COMIS 3.0 - *User's Guide*

Edited by: Helmut E. Feustel and Brian V. Smith

PREFACE

The COMIS workshop (Conjunction of Multizone Infiltration Specialists) was a joint research effort to develop a multizone infiltration model. This workshop (October 1988 - September 1989) was hosted by the Energy Performance of Buildings Group at Lawrence Berkeley Laboratory's Applied Science Division. The task of the workshop was to develop a detailed multizone infiltration program taking crack flow, HVAC-systems, single-sided ventilation and transport mechanism through large openings into account. The agenda integrated all participants' contributions into a single model containing a library of modules. The user-friendly program is aimed at researchers and building professionals.

The work was accomplished not by investigating numerical description of physicalphenomena but by reviewing the literature for the best suitable algorithm. The numerical description of physical phenomena clearly was a task of IEA-Annex XX "Air Flow Patterns in Buildings" which finished in September 1991. In Annex 23 “Multizone Air Flow Modeling,” which was adopted by the IEA-Energy Conservation in Buildings and Community Systems program in 1992, COMIS has been evaluated by means of tracer gas measurements, wind tunnel data, intermodel comparison, and comparison with analytical solutions.

From the time of its announcement in December 1986 COMIS was well received by the research community. Due to the internationality of the group, several national and international research programs were co-ordinated with the COMIS workshop. Colleagues from France, Greece, Italy, Japan, The Netherlands, People's Republic of China, Spain, Sweden, Switzerland, and the United States of America were working together on the development of the model and its evaluation.

Even though this kind of co-operation is well known in other fields of research, e.g., high energy physics, in the field of building physics it is a new approach.

This COMIS *User's Guide* contains an overview of the COMIS project as well as hints which will be useful in getting through the input and calculation procedure. The handbook comes in loose-leaf form so as to be easily updated according to the progress of the model development. Please note, that the COMIS *User's Guide* reflects the construction of the input file needed to run the calculation program COMVEN. There are several user interfaces available to create the input file and to run the program. Each interface comes with its own *User’s Guide*.

Helmut E. Feustel
COMIS Co-ordinator and Annex 23 Operating Agent
Berkeley, California
August 31, 1997

CONTENTS

1. Introduction

Infiltration models can be divided into two main categories, single-zone models and multizone models. Single-zone models assume that the structure can be described by a single, well-mixed zone. The major application for this model type is the single-story, single-family house with no internal partitions (e.g., all internal doors are open). As a large number of buildings, however, have floor plans that would characterize them more accurately as multizone structures more detailed models, taking internal partitions into account, have been developed.

Even before the advent of physical single-zone models a number of computer models had been developed to calculate the air flow distribution in multizone buildings. In these models the building is described by a set of zones interconnected by flow paths (links). Each node (zone) represents a space with uniform pressure conditions inside or outside the building and the interconnections correspond to impediments to air flow. The network models are usually based on the conservation of mass in each of the zones in the building. The first of these models to be developed was probably the BSRIA-model LEAK, which was published in 1970. Since that time many more models have been developed but most of them have been written as research tools. As a consequence they are difficult to use and are, at best, "user-tolerant" rather than "user-friendly".

Multizone models are required when there are internal partitions in a building, or, in the case of inhomogeneous concentration in the space. Multizone buildings can be either single-room structures (e.g., airplane hangars) single family houses or large building complexes. Figure 1.1 shows an example of a very simple multizone building.

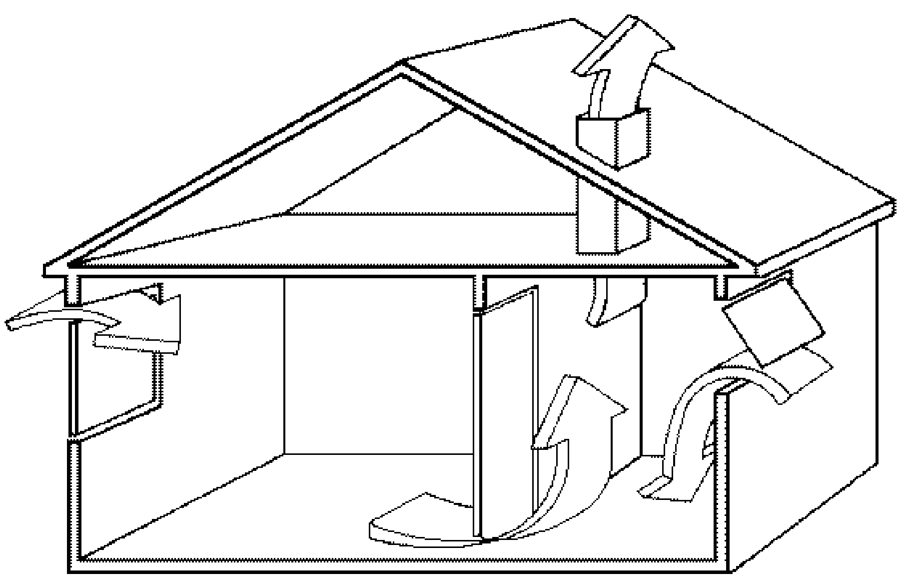

Figure 1.1: *Example of a simple Multizone Structure (Liddament 1986)*

Multizone infiltration network models deal with the complexity of flows in a building by recognizing the effects of internal flow restrictions. They require extensive information about flow characteristics and pressure distributions and, in many cases, are too complex to justify their use in predicting flow for simple structures such as single-family residences (ASHRAE 1985).

A literature review undertaken by Feustel and Kendon (1985) revealed 26 papers describing 15 different multizone infiltration models which had been developed in eight countries. A follow up review (Feustel and Dieris) carried out in 1992 produced additional information about the status of network models. One of the first to be found was Jackman's model LEAK, which was published in 1970. The latest development in infiltration modeling is the COMIS model (Feustel, et.al. 1989) which is described here in detail.

1.1 The COMIS Project

The COMIS workshop (Conjunction Of Multizone Infiltration Specialists), using a multinational team, developed a multizone infiltration model on a modular base. This model not only takes crack flow into account but also covers flow through large openings, single-sided ventilation, cross ventilation and HVAC-systems. The model contains a large number of modules which are peripheral to a steering program. COMIS can also be used as a basis for future expansion in order to increase the ability to simulate buildings.

After reviewing the available multizone infiltration models, it was clear that conventional models are not designed to be upgraded to take additional types of flow into account or to improve their usability. Therefore, we planned to design and develop a model which should contain all the missing features. From the beginning, we realized that this model development needed a lot of expertise and, equally important, manpower, which would exceed our resources.

As a consequence, we made our plan public and asked interested colleagues to join us for a twelve month period for the Conjunction of Multizone Infiltration Specialists to lay the foundation of a versatile multizone infiltration model. Almost eleven years ago, the first COMIS *Newsletter* was send to colleagues to inform them about the joint research project planned at LBL. Even though this kind of co-operation is well established in other fields of research, e.g., high energy physics, in the field of building physics it is new to engage in a research project which one individual or country would not be able to do alone. From the beginning the COMIS idea was well received. Owing to the diverse background of the group several national and international research programs are co-ordinated with the COMIS workshop.

1.2 The COMIS Model

1.2.1 Input

Special emphasis was given to the input/output routines so that the final program should be not only “user-tolerant” but “user-friendly”. It is being developed so it can be used either as a “stand-alone infiltration model” or as an “infiltration module” of a building simulation program. The input/output procedure is therefore being developed in such a way that either the COMIS input/output modules can be used or only the input/output interface. This makes it possible for the user to connect the program with other software (e.g., CAD-systems).

1.2.2 Wind Pressures

One of the major tasks has been to find a method of determining the wind pressure distribution for a building according to measured data from available literature. This allows building designers to work with the COMIS model even if wind tunnel results are not available for the building under consideration. The pressure distribution around a building is usually described by a dimensionless pressure coefficient (Cp), which is the ratio of the surface pressure and the dynamic pressure in the undisturbed flow pattern, measured at a reference height. From experience we know that wall-averaged values of Cp usually do not match the accuracy required for air flow calculation models.

In order to calculate the Cp-distribution for buildings we provide a method based on a parametrical study to determine the Cp-values. The available methods have been checked by comparing calculated results with findings from wind tunnel tests found in the literature. Since the results did not match the data well, a parametrical analysis of wind tunnel test data, aimed at developing a calculation model for Cp-data, was carried out.

1.2.3 Flow through Building Components

Crack flow, large openings and mechanical ventilation systems can be modeled by COMIS. Furthermore, additional flows which do not influence the pressure distribution in the network in a major way, i.e., simultaneous two way flow at large openings and wind turbulence effect at single-sided windows etc., were studied. Air flow rates through doorways, windows and other common large openings are significant ways in which air, pollutants and thermal energy are transferred from one zone of a building to another.

However in a previous review of multizone infiltration models, none of the described codes were able to solve this problem in any way other than to divide the large opening into a series of small ones described by crack flow equations.

COMIS's contribution to this fundamental problem was to describe the physical problem, review the various solutions developed in the literature and compare these solutions using both a numerical and a physical point of view.

The general laws demonstrated by thermal or fluid mechanics approaches are also valid for large exterior openings in steady state conditions. But none of these methods enables us to quantify the effect of an unsteady wind or large scale turbulence.

Experimental results have shown that these effects can be particularly significant in the case of one-sided ventilation. Nevertheless very few correlations have been proposed and most of those that have concern particular configurations. It seems difficult, therefore, to introduce these effects in a general way in our first model. However, we will hope to do so later on as an improvement to COMIS.

The correction of coefficients of power law for crack flow, taking into account the effect of the temperature distribution of air in the crack, is also studied. The temperature of air flowing through a crack depends on the following factors:

- air flow rate
- air temperatures of the zones on both sides of the crack
- dimensions and form of crack

In most cases the temperature of the air in a crack is quite different from the temperatures of the zones on either side of the crack. Furthermore, air leakage performance measurements are usually performed in a certain temperature condition but used at different temperatures. The temperature variation, however, has a big influence on the air leakage flow due to changes in the air viscosity and air density. Unfortunately, almost all the models dealing with air leakage characteristics ignore this phenomenon.

Data obtained from measurements on crack models show that, for turbulent crack flow, the mathematical description of the friction factor is identical with the one found for conduit flow with smooth walls. Therefore crack flow can be seen as duct flow with a more complicated flow path.

We found from the crack flow equation research that the flow performance is strongly temperature dependent. In order to arrange the results in the usual form we have introduced correction factors which account for the temperature influence. The correction factor depends on the type of leakage. We have developed three different equations for the different correction factors.

We can easily build an air leakage temperature module according to the crack forms. Fortunately, we found that the crack form mainly depends on the structure of the building or on the type of building component and that its size depends on the workmanship. We therefore classify crack forms into three groups: double frame windows, single frame windows and doors, and walls.

HVAC-Systems (Heating, Ventilating and Air-Conditioning Systems) are composed of ducts, duct fittings, junctions, fans, air filters, heating and cooling coils, air-to-air heat exchangers, flow controllers, etc. Several of the program modules concerning ventilating systems have already been developed, allowing us to calculate the coefficients of the flow equation for duct works with fittings, the static pressure losses for T-junctions and the volume flow rate of a fan as well as for a flow controller as a function of the pressure difference. Since the duct systems are described by a network in the air flow model the junction is treated as a pressure node.

There are some data available in the literature for the pressure loss coefficients at the T-junction. To our surprise we found that the values of the pressure loss coefficients were significantly different for different sources. For example, in the case of converging flow, the pressure loss coefficient through the main duct of the T-junction obtained from one source is double the value of the loss coefficient given in another.

The fan performance curve is expressed on the basis of more than three data sets of the volume flow rate, and the pressure difference by the polynomial approximate formula using the least square method.

The pressure loss curve for flow controller is expressed by equations based on data sets of the pressure loss and the volume flow rate. The input data is the driving pressure difference of the flow controller. The output data is the volume flow rate.

There are other components connected to the HVAC-Systems which cause dynamic pressure loss, e.g., air filters, heating or cooling coils, different types of junctions, etc. The calculation procedures for these components will be added as soon as possible.

1.2.4 Solver

Calculating the infiltration and ventilation flow rates requires the solution of a non-linear system of equations. The main task has been to find an efficient solving method.

A building is basically modeled by pressure nodes that are interconnected with air flow links. For one time step, the outside of the building is represented by a fixed boundary condition. The pressures of the internal nodes in the air flow network have to be solved so as to determine the different air flow rates. Solving these infiltration and ventilation flow rates requires the use of a non-linear system of flow equations. The main task was to find an efficient and stable method.

The starting point is the Newton-Raphson method, with derivatives, operating on a node-oriented network which, in most cases, quickly brings about the convergence of the system of equations. The method has been modified to avoid occasional convergence problems when working with power functions. Fortunately, the origin of the convergence problems is well understood. The solving method works on the flow balance equations and not on the flow equations. If one or several of these balance equations have an exponent close to one-half, the Newton-Raphson method will not work well, due to the nature of the procedure, in finding the next approximation. One instance when this happens is when a leakage opening with a flow exponent of one-half is predominant in one zone. In this case the flow balance equation will also have an exponent close to one-half. An under-relaxation will increase the convergence velocity and bring us to the solution. In principle it is a question of finding an appropriate relaxation coefficient.

1.2.5 Follow-Up of COMIS

Work on the COMIS program did not finish by October 1989. A computer code was available but such a program is ever perfectible. The validation procedure itself is a huge work and was not completed during the COMIS year. Moreover, new knowledge (e.g. from Annex XX) was available after the COMIS workshop finished. This knowledge had to be integrated into this program.

A roundtable discussion between the COMIS participants and the COMIS review panel about future perspectives revealed a strong feeling that COMIS (or its successor) ought to operate as an international institution with participants committing themselves to a definite work load.

The COMIS group suggested to start a working group under the hospice of IEA's Energy Conservation in Buildings and Community Systems program. The *Multizone Air Flow Modeling* working group was officially adopted in June 1990 as Annex 23.

The objective of this annex was to study physical phenomena causing air flow and pollutant transport (e.g., moisture) in multizone buildings and to develop modules to be integrated in a multizone air flow modeling system. Special emphasis was to the comparison between results from the model and from in-situ tests.

To reach these objectives the project is structured in three subtasks:

- System development (subtask 1)
- Data acquisition (subtask 2)
- System evaluation (subtask 3)
- Results of these subtasks are addressed to researchers and consultants and will promote an energy efficient design. The Participants undertook a task sharing project involving model development, data acquisition and analytical studies.

A close cooperation was envisaged, mainly with regard to state-of-the art reviews, data collection, coordination of work, e.g., defining cases for evaluation purposes with other pertinent projects. The Air Infiltration and Ventilation Centre will, as part of its on-going work plan, act as a vehicle for disseminating the results of this particular Annex. A data base for evaluation purposes is going to be prepared by AIVC. The Centre has already started to collect wind pressure data and leakage data. Algorithm developed by Annex XX were incorporated into the modeling system. Data obtained for evaluation purposes by this annex was used for subtask 3. The overlapping in time with Annex XX guaranteed a sufficient transfer of knowledge.

There are several documents published (or will soon be published) by COMIS workshop and Annex 23 participants:

- User's Guide for COMIS and IISiBat/COMIS
- COMIS Fundamentals (Feustel and Raynor-Hoosen 1990)
- Special Issue of the Journal Energy and Buildings (Feustel 1990)
- Special Issue of the Journal Energy and Buildings (Feustel 1997)
- Evaluation of COMIS (Fürbringer et.al. 1996)
- Evaluation of COMIS - Appendicies (Fürbringer et.al. 1996)
- Programmer's Guide (Dorer and Weber, 1997)

1.2.6 References

Dorer, V and A. Weber: "COMVEN 3.0 Programmer's Guide," EMPA Dübendorf, 1997

Feustel, H.E. (Editor): Special Issue of the Journal Energy and Buildings on Air Flow Modeling Issues, Energy and Buildings, to be published in 1997

Feustel, H.E. (Editor): "The COMIS Model," Special Issue of the Journal Energy and Buildings on Air Flow Modeling Issues, Energy and Buildings, Vol. 18, No. 2, 1992

Feustel, H.E. and J. Dieris: "A Survey of Air Flow Models for Multizone Structures," *Energy and Buildings*, Vol. 18, 1992

Feustel, H.E. and A. Raynor-Hoosen (Editors): "Fundamentals of the Multizone Air Flow Model -COMIS," Technical Note AIVC 29, Air Infiltration and Ventilation Centre, Bracknell, UK, 1990

Feustel, H.E., F. Allard, V.B. Dorer, M. Grosso, M. Herrlin, M. Liu, J.C. Phaff, Y. Utsumi, and H. Yoshino: "The COMIS Infiltration Model", in Proceedings, "Building Simulation '89, The International Building Performance Simulation Association", Vancouver, 1989

Feustel, H.E. and V.M. Kendon: "Infiltration Models for Multicellular Structures: A Literature Review," *Energy and Buildings*, Vol. 8, No. 2, 1985

Fürbringer, J.-M., C.-A. Roulet, R. Borchiellini: "Evaluation of COMIS," Swiss Federal Institute of Technology, Lausanne, 1996

Fürbringer, J.-M., C.-A. Roulet, R. Borchiellini: "Evaluation of COMIS - Appendices," Swiss Federal Institute of Technology, Lausanne, 1996

Jackman, P.J.: "A Study of Natural Ventilation of Tall Office Buildings," Inst. Heat Vent. Eng., Vol 38, 1970

Liddament, M.W.: "Air Infiltration Calculation Techniques - an Applications Guide," Air Infiltration and Ventilation Centre, Bracknell, 1986

ASHRAE 1985 Handbook of Fundamentals, Chapter 22, American Society of Heating, Refrigerating and Air-Conditioning Engineers, Atlanta, GA, 1985

2. How to use COMIS

COMIS can be used either with a user-interface (e.g., IISiBat, COMERL), or as a stand-alone program without a user-interface. In the latter case, the input file either already exists or will have to be developed with a text editor. The procedures described here are valid for the installation of the program without an interface. For program usage with an interface, please refer to the user's guide provided by the interface developer.

2.1 Installation

The COMIS distribution diskettes contain the MS-DOS executables and sample input files. To use COMIS under a different operating system one must obtain the COMIS source code and recompile the programs for that system.

For installation on a PC first create a directory on your hard disk where you would like to install COMIS. For example, to install COMIS on drive "C", type:

MKDIR C:COMIS

Next, insert the first COMIS disk in the 3.5" drive and type:

COPY B:*.* C:COMIS

This will install the files into the directory COMIS on your hard disk. Now insert the second COMIS disk in the floppy drive and repeat the "COPY" command.

You must not be running Microsoft Windows® when running COMIS because of memory limitations.

If you are working with the pre-compiled versions of COMIN and COMVEN you should skip ahead to 2.2.

2.1.1 Compilation of COMIN

COMIN is not any longer being supported.

2.1.2 Compilation of COMIS

The COMIS source is subdivided into more than 40 files. There is a Makefile for automatically compiling COMIS on a Unix system. For compiling COMIS on a PC, compile all of the ".F" files then link into one ".EXE" file. COMIS has been successfully compiled on PCs using the NDP FORTRAN compiler from Microway and the FORTRAN 77 compiler from Watcom. COMIS is a large program which cannot be compiled and run with ordinary FORTRAN compilers. NDP FORTRAN allows programs to be larger than 640k bytes by having its own loader to load the program into memory. This is accomplished by linking COMIS with the -bind option which creates the COMIS.EXE file with its loader in it.

2.2 How to get Started

To start working with COMIS it is recommended to **read this handbook carefully** and then to run one of the user interfaces (COMIN, COMERL, or IISiBat) in order to establish a first version of an input file.

Originally, there were two programs which make up COMIS. These were COMIN and COMVEN. COMIN is being succeeded by several user-interfaces available for Pcs and workstations, while COMVEN has been renamed to COMIS.

- Typing **COMIS** starts the modeling program

Called this way, the program uses the specifications for input and output files which are set in the COMIS.SET file (which is described below), however, the better way to run COMIS is to give the names for the input and output file on the command line. In this case, no COMIS.SET file is needed. The commands would be the following:

for Unix or the PC:

COMIS -i *input file name* -o *output file name*
-t *project name*
-a *new file name*
-uu *unformatted output file name*
-uf *formatted output file name*

The commands 'COMIS -uu name' and 'COMIS -uf name' allow the user to create his own formatted or unformatted output file. This option allows to manipulate and/or to reduce output for absolute necessary data. The output routine has to be modified by the user according to his needs. For more details, please see Section 7.2.2. This option is only available with user-compiled programs

The command 'COMIS -a' gives the files a new prefix name. E.g., the user would like to name the files after the name of the project, then one enters 'COMIS -a richmond'. In this case all files that are mentioned in the set file get new prefix names :

richmond.cif	or	richmond.cif
richmond.cof		richmond.uof

If COMIS is called without any options and if no specifications for input and output files are made in COMIS.SET then the default filenames are taken (see also description of COMIS.SET below).

2.3 Description of the COMIS.SET File

The COMIS.SET file can be used to describe conversion units (e.g., C for temperatures, Pa for pressures, etc.) used for COMIS and the default input and output file to use. Its use for describing unit conversions is discouraged and only remains for backward compatibility for older .SET files that users may have.

A much better way to define conversion units is to use the &-PR-UNITS keyword in the input file itself. Because the description of the units are now in the input file, it is clear which units were used for the simulation.

The COMIS.SET file consists of three parts: The print-level selection part, the part for the input and output conversion units and the part for specification of input and output files. An example for a COMIS.SET file is given in Figure 2.1.

Before a run with any part of the program is made one should be sure that the settings in COMIS.SET are correct for the run. To give some help an explanation of the various parts of the file is following.

- **First Part: print-level selection values**

The lines have to be in the following sequence and it is not allowed to omit any lines. The higher the value that is used for lines 3 to 6 the more output is produced.

1. Line: 0= write output in output file (*.COF file)
   1= write output on the screen
2. Line: 0= no additional output for program test runs

1= additional output for program test runs

3. Line: print-level for input; range: 0..20
4. Line: print-level for precalculation; range: 0..20
5. Line: print-level for solver; range: 0..20
6. Line: print-level for output; range: 0..20
   0 = no output
   1 = link data only
   2 = zone pressures and link data

- **Second Part: Input and Output Conversion Units**

The first two lines of this part tell how many input and output conversion units respectively follow. The default values are 10 and 13. The program reads the units in a fixed order, always starting with the unit for air leakage. Below the units for both input and output are listed in the sequence from 1 to 13. Note that the last three units (air change rate, mean age of air and energy) are only used for output unit conversion. The input units must be first, followed by the output units.

NOTE: All the values of the following types in the input file are converted by COMIS to the default values. An example for a type not being changed is the barometric pressure. Input values for barometric pressure have to be always in kPa.

Line	Name	Default Unit	Other possibilities
1	Air Leakage	Cm(kg/s@1Pa)	kg/h@1Pa, kg/s@10Pa, kg/h@10Pa dm3/s@1Pa, m3/s@1Pa, m3/h@1Pa, dm3/s@10Pa, m3/s@10Pa, m3/h@10Pa, ELA4, ELA10
2	Ventilation Mass Flow	kg/s	g/s, g/h, kg/h, m3/s, dm3/s, dm3/h, m3/h
3	Ventilation Pressure	Pa	mmH2O, mmHg, inH2O, inHg, hPa, kPa
4	Temperature	C	K, F, Ra, Re
5	Humidity Concentration	g/kg	kg/kg, mass%
6	Pollutant Source	kg/s	g/s, g/h, kg/h, m3/s, dm3/s, m3/h, dm3/h, ug/s
7	Pollutant Sink	kg/s	g/s, g/h, kg/h, m3/s, dm3/s, m3/h, dm3/h, ug/s
8	Pollutant Concentration	kg/kg	g/kg, mg/kg, ug/kg, m3/kg, dm3/kg, ml/kg, ul/kg, mass%, ppm, ppb, part (any rational combination of g, kg, mg, ug, m3, dm3, ml and ul are possible, e.g. mg/g, dm3/mg)
9	Fan Flow	m3/s	g/s, g/h, kg/s, kg/h, dm3/s, m3/h, dm3/h
10	Wind Speed	m/s	cm/s, kt, ft/min, km/h, mile/h
11	Air Change Rate	1/h	1/s
12	Mean Age of Air	h	s
13	Energy	KWh	J

- **Third Part: Input and Output Files**

In the last part of COMIS.SET certain keywords for the various parts of the COMIS program are used. The keywords are followed by the names for the input and output file. If they are not written then some default file names are taken. Each keyword must start with a '&' sign in the first column. Shown below are the three different parts of the program and some further information on input and output.

&-COMIS:

INPUT:	name for the input file, default name is 'COMIS.CIF'
OUTPUT:	name for the COF file, default name is 'COMIS.COF'
TABLES:	Here the project name, which is used for the names of the table files, can be defined. The table created may later be processed with any spreadsheet program. Only the numerical results are printed in this file, separated by the separator strings specified. This separator can be any sequence of ASCII characters (including an empty string) and must be enclosed within quotes ("). Examples of separator strings are: "/" ; ";" ; " " ; "" ; "*"

Two combinations for the TABLES input line are possible:

Examples of input lines:	files created with COMIS
TABLES ";" NAM2	xxNAM2.CSO
TABLES ";"	xx.CSO

The names of the output files consist of two parts: xxyyyyyy.CSO
where:

xx	Prefix 2 characters: Depending on what data the file contains, e.g. PZ for pressures in zones. Which files will be created with COMIS is set in CIF (Section 5.2.2)
yyyyyy	Project name 0...6 characters: Defined here under keyword TABLES.

The appearance of the keywords **&-COMIS** in the third part of the COMIS.SET file is mandatory.

There is no output unit for the fan flow specified, as the fan flow is being displayed as a "link" in the units of ventilation mass flow (see Chapter 7).

```
00 (1=write(CRT.*) on CRT; 0=write(CRT.*) on COF (output file))
00 (print-level for test runs. Range 0,1)
02 (print-level for input. range 0,20)
02 (print-level for precalculations. range 0,20)
03 (print-level for solver. range 0,20)
05 (print-level for output. range 0,20)
10 (number of user units for input)
13 (number of user units for output)
kg/s@1Pa    ( input userunit for  air leakage            Cm =flow kg/s @ 1
kg/s  ( input userunit for  ventilation   massflow  kg/s )
Pa    ( input userunit for  ventilation   pressure  Pa   )
C     ( input userunit for  temperature             C    )
g/kg  ( input userunit for  concentration humidity  g/kg )
kg/s  ( input userunit for  pollutant source        kg/s )
kg/s  ( input userunit for  pollutant sink          kg/s )
kg/kg ( input userunit for  pollutant concentration kg/kg)
m3/s  ( input userunit for  flow through fan        m3/s )
m/s   ( input userunit for  wind velocity           m/s  )
Cm    (output userunit from air leakage             Cm   )
kg/s  (output userunit from ventilation massflow    kg/s )
Pa    (output userunit from ventilation pressure    Pa   )
C     (output userunit from temperature             C    )
g/kg  (output userunit from humidity                g/kg )
kg/s  (output userunit from pollutant source        kg/s )
kg/s  (output userunit from pollutant sink          kg/s )
kg/kg (output userunit from pollutant concentration kg/kg)
m3/s  (output userunit from flow through fan        m3/s )
m/s   (output userunit from wind velocity           m/s  )
1/h   (output userunit from air change rate         1/h  )
h     (output userunit from mean age                h    )
J     (output userunit from energy                  J    )
&-COMIS
 INPUT TEST.CIF
OUTPUT COMIS.COF
TABLES xy "   "
```

Figure 2.1: *Example of a COMIS.SET file*

3. Input

There are several interactive input programs which may be used to create or modify an input file for COMIS. These are COMIN, COMERL and IISiBat. While COMERL and IISiBat have their own User’s Guide, COMIN is not any longer supported.

The following chapters explain the structure of the COMIS input file. The input file might either be developed by any of the user-interfaces or by using a text editor.

3.1 Non-Interactive Input

The purpose of the interactive input programs is to establish an input file which is subsequently read by the program COMIS. This input file can also be established using the screen editor. The structure and the format of the input file are described in more detail in Section 4.4. and 5. An example is given in the appendix.

The minimum requirements for the contents of an input file are:

- Data section keyword
- Data per data block according the description given in Section 5.
- Data must be given in the prescribed order of the data blocks .

Headers may be omitted in the input file but in this case data must be given starting with the first data block. If headers are included the earmark sign in the upper left corner of the header gives the data block number.

Minimum data requirement for air flow component input:

- keyword '&-NET-AIR'
- all component data must be under this 'main' keyword
- keyword according to selected component type
- Data according to description starting with data for the first header. Headers may be omitted.

4. Input File

4.1 Purpose

The input file is the only data interface between the user-interfaces and the simulation program COMIS. There are three ways to establish the input file:

- Using an interactive input program
- Using the screen editor
- Using both, interface and screen editor

4.2 File Name

The input file name extension must be '.CIF'.

4.3 Data Structure

4.3.1 General Classification

The following classification is used throughout this guide:

- Data group:
  e.g., Problem, Network, Schedules...
- Data section:
  characterized by a &-keyword in the input file
  Example: &-PR-IDEN problem identification
- Data block:
  data under one specific data header; the block can contain several data lines
- Data set:
  data under one specific *name/*no (zone layer, schedule, etc.)

4.3.2 Data Sections

The input file is divided into data sections. The beginning of each section is characterized by a specific keyword, and these are characterized by having an '&' in the first column (Example: &-PR-IDENTification). Within one data section there may be more than one header per data set.

4.3.3 Headers

Data blocks may be preceded by the respective header line set containing the parameter names, default units and possibly some further descriptions. This allows for the content of the data file to be self-explanatory. These header lines (but not the keywords!) may be omitted by the user while establishing an input file using the screen editor. Units reported in the header correspond to those units used internally by COMIS. The units used in a particular input file have to correspond with the setting in the COMIS.SET file or in the input file!

Headers are surrounded by vertical and horizontal bars and show the kind of data needed in the data-lines beneath them. If a header has a number in the upper left corner then those numbers have to be in order and also the corresponding data-lines. If headers are not in the right order, the program will mix up the data lines. Headers are not repeated for every data line.

4.3.4 Data Separator

The separator for data items is a space, a comma or both. A data-line is not allowed to start with a comma. Data-items in one data line must be in sequence but they do not have to be in a certain column and also not straight under the header. It is, however, better to use the header as a formatting aid.

4.3.5 Special Characters

The "#" and the more commonly used ";" characters mean: comments follow in this line. A "#" inserted in the first column of a data line converts this line to a comment. This makes it easy to change a line by copying it into a new line and yet keeping the old line with "#" before the first data element

The special characters used are:

&	(column 1) A keyword follows in this line
#	(column 1) Comment follows on this line
|	the headers are surrounded by bars '|' and under-scores '_', (ASCII char 95) the first line of a header is, preferably, close under the KEYWORD and starts with two blanks and then a row of underscores '___'
*<name>	<name> being the name of an airflow component, schedule or zone in schedules, zone-layers, pollutants.

4.3.6 Required Data / Optional Data / Defaults

The type of brackets used for the units in the headers indicates whether the specific parameter is required or optional:

(..)	Required parameter or set of parameters
[..]	Optional parameter or set of parameters
( ) ( ),( ) ( )	means: connected pairs of data

Optional parameters in an input line are not needed but if they are between mandatory parameters then something must be there in their place. Only the optional parameters following the last mandatory parameter may be omitted entirely.

Any abbreviation of the word "default" (without a period ".") may be used in a place where the user wants to use a default value, or to skip over an optional parameter. Examples are "d", "def". Uppercase and lowercase are not important.

4.4 Example File

An example of an input data file is given in Section 6.

INPUT FILE

PROBLEM DESCRIPTION

DIRECT NETWORK DESCRIPTION
- AIRFLOW COMPONENTS
- HVAC, TRANSITION, CRACK-TEMP DATA
- ZONES
- NETWORK LINKS

SCHEDULES
- MAIN SCHEDULES
- LINK SCHEDULES
- ZONE SCHEDULES/ OCCUPANT SCHEDULES
- INPUT FOR SCHEDULE GENERATOR MODULE

CP VALUES

DIRECT Cp VALUES INPUT

CP VALUE CALCULATION ROUTINE
- BUILDING ROUGH OUTSIDE SIZE
- FACADE ELEMENT 2D-POSITION DATA
- WIND DIRECTIONS

ENVIRONMENT DESCRIPTION
- BUILDING RELATED PARAMETERS
- WIND AND METEO RELATED PARAMETERS

METEO DATA

POLLUTANT DATA
- POLLUTANT DESCRIPTION
- OUTDOOR POLLUTANT SCHEDULE

OCCUPANT DESCRIPTION

5. Input Data Description and Input Format

5.1 General

5.1.1 Introduction

In this chapter all input parameters are described in detail. The description of the parameters is structured according to the sequence of data sections and data blocks given in the input file.

- Problem description
- Air flow components
- HVAC
- Zones
- Links
- Schedules
- Cp-Values
- Environment description
- Meteo description
- Pollutant description
- Occupant description

The following chapters will show the headers, explanations of the input, and some example inputs. The example inputs are shown in italics characters.

5.1.2 Time and Date Format

The input format for time and dates is given in this section because time data might be needed in several data sections.

As shown here the date can be entered using different formats:

MDD_	MMDD_	YYMMDD_	YYYYMMDD_
janDD_	janDD_	YYjanDD_	YYYYjanDD_

in which M=a digit of the name of the month
D=a digit of the calendar day number
Y=a digit of the year number
jan=ja; jan; januar; january; fe; feb; februa;
mar; apr; may; jn; jul; aug; sep; oct;
nov; dec.

In the case the name of the month being read into the program, any valid and unique fraction of the month name is satisfactory. This part of the input is case insensitive, so "JAN", or "Jan", for January would work, too.

The formats for the time input are:

h	hh	hmm	hhmm	hmmss	hhmmss h:
	hh:	hmm:	hhmm:	hmmss:	hhmmss:

in which h=digit of the hour
m=digit of the minute
s=digit of the second

The date time combination must not contain any blanks!

Following are some examples of how the data for dates and time might be entered into the program:

89aug04_18:23:10	or	*890804_18:23:10*		
1989aug04_18:23:10	or	*19890804_18:23:10*		
aug04_18:23:10	or	*0804_18:23:10*	or	*804_18:23:10*

Examples of the date_					interpretation
*89aug04_*	or	*890804_*			1989 august 04
*1989aug04_*	or	*19890804_*			1989 august 04
*aug04_*	or	*0804_*	or	*804_*	august 04 any year

Instead of a "date_" expression a weekday name is also valid. Examples are

Mon_	or	MON_	or	Monday_
Fri_	or	FRI_	or	Friday_

It is also possible to use the expressions Weekday (WDY_) or Weekend (WKD_).

Examples of the time			interpretation
18:23:10	or	*182310*	18 hour 23minutes 10seconds
18:23	or	*1823*	18 hour 23minutes 0seconds
18:	or	*18*	18 hour 0minutes 0seconds

In order to distinguish between the date and the time, the date is followed by an underscore character '_'.

5.1.3 Data from Files

In many data sections the user may define a data file instead of giving explicit input data. The format of this file allocation is:

F : <filename>

The same command can be used to connect a file which contains data for schedules (see specific Sections):

F: <schedule name> <filename>

where <filename> can be any file designation in compliance to the specific computer operating system requirements.

Example of a Cp-value data file assignment:
for DOS-System: F:C:\comis\exampl\comin\cp1.dat

for UNIX-System: F:~comis/@exampl/comin_cp.cp1.dat
for VMS -System: F:[username.comis.exampl]comin_cp_cp1.dat

The content of these files must be formatted exactly according to the requirements set up in the following sections for the specific data.

5.2 Problem Description

5.2.1 Problem Identification

Keyword:

&-PR-IDENtification

Header:

1.	Problem name

Example input:

This is an example of the problem name.
You may write here text to describe the problem case.
The maximum number of lines is 10.

Description:		
Parameters	Description:	Input Format
Problem Name	max. 10 lines of description	string lines max. 10 lines with max. 79 characters each

Header:

2.	**Versionname**

Example input:

24-June-89

Description:		
Parameters	Description	Input Format
Version name	---	string line

5.2.1.1 Problem Input/Output Units

Unit conversion factors as shown here can be used at all data sections. The data section &-PR-UNITS can be used more than once. This allows having different units in different data sections within one input file. The units declared by &-PR-UNITS are valid for all data sets following this declarations until the end of the input file or until another unit-declaration is provided.

Keyword:

&-PR-UNITS

Header:

Unit Conversion Definitions		
Name	Input	Output

Example input:

Airleakage	kg/s@1Pa	kg/s@1Pa
massflow	*kg/s*	*kg/h*
pressure	*Pa*	*Pa*
temperature	*C*	*F*
humidity	*kg/kg*	*kg/kg*
source	*kg/s*	*kg/s*
sink	*m3/s*	*m3/s*
concent	*kg/kg*	*ug/m3*
fan	*m3/s*	*m3/s*
velocity	*m/s*	*kt*
INPUT		
Profile	*alpha*	
OUTPUT		
ach		*1/h*
meanage	*s*	
energy		*kwh*

The unit conversion factors are used at all data sections that follow this input. The **&-PR-UNITS** keyword may be used more than once. This way, different parts of the input file may have different units! The **&-PR-Units input** is the preferable method of defining the unit conversions because a simulation situation is fully contained in the input file. This contrasts with using the COMIS.SET file to define the units, where the input units are disconnected from the input file. In the latter case it is impossible to tell which input units were used from looking at an output file.

If the keyword INPUT is seen, the following lines of units are only used for input conversions. Conversely, if the word OUTPUT is seen, the units are used only for output conversions.

If neither INPUT nor OUTPUT is seen, the first value following the unit name is for input conversions and the second for output.

The air change rate (ach), mean age of air (meanage) and energy units are only available for output conversion and **must follow the OUTPUT** keyword. All the other units may be used for both input and output conversions.

Only 6 letters are significant, but more may be entered; e.g. "temperature."

The units are fully described in Section 2.3 - Description of the COMIS.SET file.

Interpretation of zone pressures

COMIS normally uses pressures with reference to the outside pressure (without wind) at the reference level of the building. This means that the pressure decreases about 12 Pascal (Pa) per meter height, as in reality. This is due to the gravity field. Each m^3 of air weighs about 12 N, which results in the 12 Pa/m pressure change.

Consider two rooms, one on top of the other (**Figure 5.2.1**).

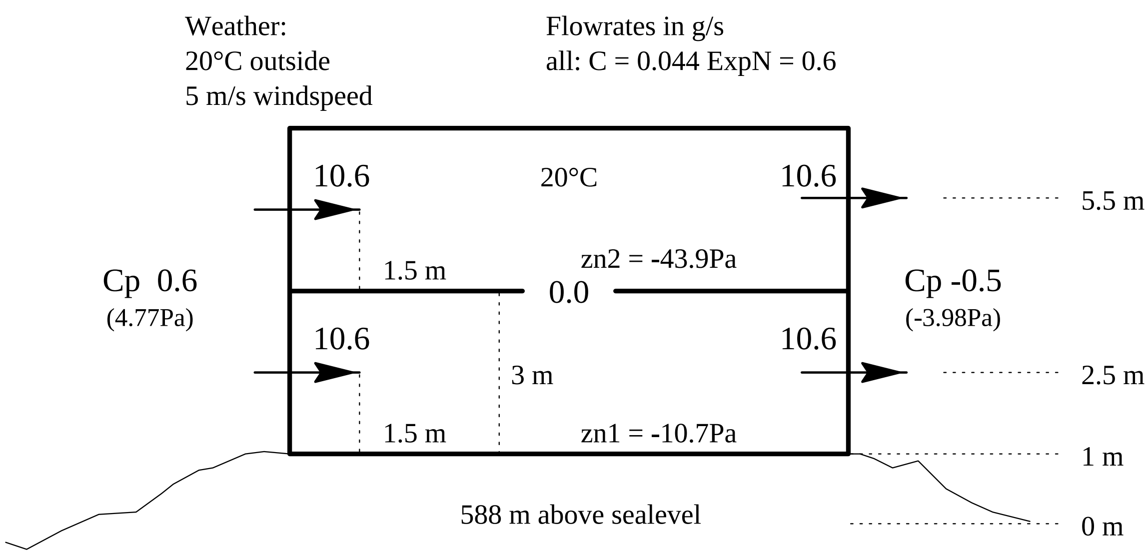

Figure 5.2.1: *Two zones with normal pressure output.*

If there is no flow through the cracks of the ceiling between the rooms, then the pressure in the upper room is about 36 Pa lower than in the lower room. Here it is about 33 Pa due to the altitude of the building of 588 m. Both room pressures (-10.7 and -43.9 Pa) are defined here at the floor level and the building reference level is one meter below the floor of zone 1. Looking at the zone pressures, in Figure 5.2.1, one might first think that there must be a flow from zone 1 to zone 2. However this flow is zero. At a height of 3 m in zone 1 the pressure equals the pressure in zone 2 at 0 m.

One could say: as there is no flow between the two rooms their pressures are equal, therefore they must be the same in the COMIS output. A way to do this is simple: use as reference pressure the outside pressure (without wind) at the altitude of the zone reference height (**Figure 5.2.2**).

Zone pressures with reference to outside pressure
at respective zone reference height; Tin-Tout=0°C

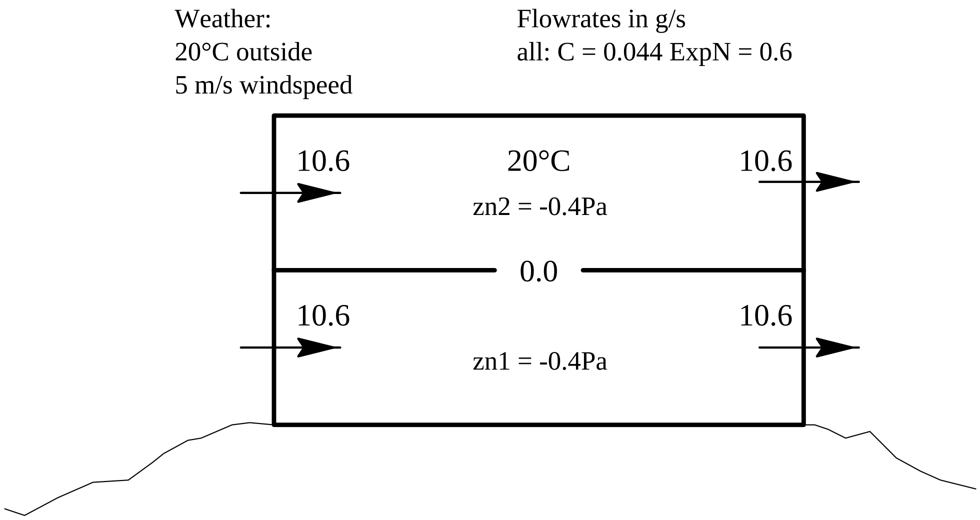

Figure 5.2.2: *Same as Figure 5.2.1, but with outside pressures at zone reference height as reference.*

The pressures now almost resemble the pressure hierarchy. However if the rooms have a temperature that differs from outside, the thermal stack pressure (with respect to outside) can still result in a small pressure difference between the rooms that seems not to agree with the flow direction at first glance (see **Figure 5.2.3**).

Zone pressures with reference to outside pressure
at respective zone reference height; Tin-Tout=10°C

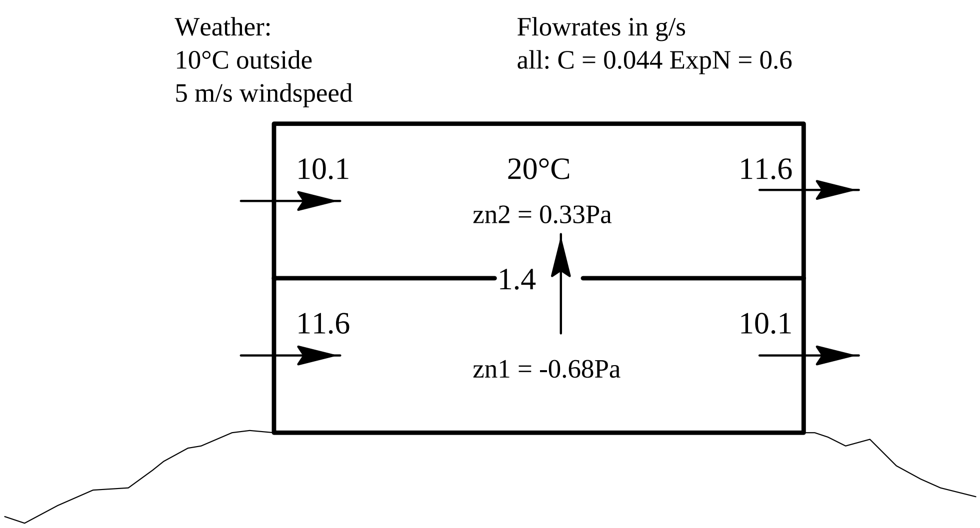

Figure 5.2.3: *As Figure 5.2.2, but with a 10K stack pressure difference.*

If the characters '-OSR'(take “Outside Stack as Reference”) are found attached to the output units for pressure (e.g. "Pa-OSR") then the corrected zone pressures are printed.

5.2.2 Problem Simulation Options

Keyword:

&-PR-SIMUlation options

Header:

Simulation Option Keywords: One keyword per line Keywords may be preceded by “NO”		
VENT:ilation CONC:entrations	POL:lutant	HEAT:flow‡
	INPUT echo DEFAULT echo SET echo UNIT	
SCHED:time <time> START:time		

Example Input:
VENTilation
NO HEATflow
POLlutant 1,2
DEFAULT echo

STARTtime 1988jun01_10:00
Stoptime 1988jun01_11:00 keep

‡COMIS 3.0 only allows the calculation of ventilation and concentrations.

Keyword Format	Description

```
XXX:xxx
|     |
|     This part may be added to the keyword
+---  This part has to be typed exactly as given
```

Each keyword may be preceded by the word 'NO' thus canceling the keyword. Only one keyword per input line. Keywords may be given in any order.

VENT:ilation	Calculation with COMVEN
HEAT:flow	Calculation with heatflow module (if available)
POL:lutant (<No>)	Pollutants are taken into account. If a No is specified, only Pollutant No <No> is considered for output. Default : all
CONC:entration	Concentrations per zone for specified pollutants
INPUT echo	Input file is printed in .COF
DEFAULTS echo	Default values are specified in .COF; provides three levels (1..3)
SET echo	SET-file is printed in .COF
UNIT	Print all input/output units used by COMVEN
SCHED:time <time>	Start time of processing the schedules
START:time <time> [CONT|REUSE]	Start time of simulation
STOP:time <time> [KEEP]	End time of simulation <time> is the date-time definition string (see Section 5.1.2) CONT after the start time means an old TMS-file has been kept and shall be reused, with the start time of the new run equal to the stop time of the previous run. REUSE stands for using the whole TMS-file of the last run again, with the same start and stop time and unchanged length of the input file(s). KEEP after the stop time means that the TMS-file shall be kept after the simulation is completed.

Start and Stop Time of the Simulation

Start and stop time of the simulation should be written under the keyword &-PR-SIMU options. It is possible to have a start time, which is equal to the stop time. That means only one time-step is performed. It is not allowed to have a stop time earlier than the start time. In this case the program stops with the error message 'Error in Input File: Stop time less than start time.'. If no times are given, the program will search the **&-SCH-METeo** section for a time span. This will only work correctly, if the full time description is being used.

All schedules will be searched for the event at or before simulation start time. If no value is found, default values will be taken.

A third keyword handled by the time management system (TMS) is the keyword SCHED, which specifies the time the program starts to read and process the schedules. The schedule time may be later than the start time to allow some fixed pre-conditioning and delay the normal operation of the schedules, and is equal to the start time as a default. The SCHED keyword can be important, e.g., if one wants to split a run over one year into 4 runs simulating three months each. If one wants that the second run starts with the same conditions the first run ends, a schedule time equal to the start time of the first run can be specified. In order to save some time the TMS file of the last run can be reused again which saves time that would be necessary for creating and sorting the schedule data. In this case the start time of the following run must be the same as the stop time of the preceding run for which the TMS file was created and kept for later use (CONT keyword).

To tell the program that the TMS file from a former program run shall be used the CONT keyword or the REUSE keyword must be specified behind the start time. IF the REUSE keyword is used, the whole TMS file from the last run is used again. This option must be used carefully. It does not work if a line in the input file or in a schedule file has been inserted or deleted. Furthermore, start and stop time have to be the same as in the previous run. The values of schedule-data or other input values, however, can change as well as the weather file. If one wants to keep the TMS file of the current simulation run for later use, the KEEP option must be used at the end of the stop time definition.

NOTE: In older versions of COMIS the &-PR-OUTP section also contained the simulation information which is now in &-PR-SIMU. For compatibility with older input files both sections may appear under the &-PR-OUTP keyword.

Defaults in COMIS.

COMIS has default values for most variables read from the input file. Most defaults are assigned in the main program. From each input line that contains data, COMIS reads a programmed number of variables. As soon as there is no more data in the line, and the program still tries to read data, the variables are filled with their defaults.

The data in the input line must be in sequence. we do not know a value for a required input we could use the default. This is done by any of the following data entries: 'd' 'de' 'def' ... 'default' , as seen in the example.

COMIS can report all defaults it uses when reading an input file. This can be a long list. The Keyword to echo all defaults is the word DEFAULT and must be given at &-PR-OUTPut options.

For example, the default input echo might look like this:

&-NET-LINks

```
Default value   1.0000     used for HfL(Nl)  in line
 l1     TD1    -1   z1      3   3      d        50
```

5.2.3 Problem Output Options

Keyword:

&-PR-OUTPut options

Header:

Output Option Keywords: One keyword per line		
Keyword {Link/Zones}		
Define data to be stored (append -S for Storing each value):		
PZ {Zones} = Pressure/zone	FL {Links} = Flow/link	HZ {Zones} = Humidity/Zone
TZ {Zones} = Temp./zone	TL {Links} = Temp./link	IZ {Zones}= Infil/Zone
FZ {Zones} = Flow/zone	SL {Links} = Status	AZ = ACH
WA = Wind Velocity	HA = Outdoor Humidity	MZ {Zones}= Age of air/Zone
Cn {Zones} = Concentr.	TA = Air Temp.	EZ {Zones}= Ach index/Zone
Sn {Zones} = Poll. Sink	Qn {Zones} = Poll. Source.	
for Gas n (1<= n <=5)	PE {Points} = Wind pressure	
FB = Flow matrix/building	IB = Outdoor infil/building	AB {Zones}= Outdoor ach/building
MB = Arithmetic mean of building mean age of air		
RB = RMS of building mean age of air		
NB = nominal time constant of building mean age of air		
EB = ACH efficiency of building		
LB = Ventilation heat loss energy of building		
For mean values replace -S with -T		

Example input:

PZ-S 1-10,38,100-115,200
PZ-T 1-5,4-7,6-10,10,38,100,101
PZ-T 200
TZ-S 1-20
TZ-T 3-5,5,6
TA-S
C1-T 1,2,3

The keywords for creating output files (*.CSO files) that may later be used by a spreadsheet program consist of two letters (or one letter and one digit) plus either `-S' or `-T' (**S**tore each value or store mean values for the **T**otal simulation period). This works only, if the entity covered by the keyword allows a parameter (such as the list of zones in `PZ-S 1-10'=`**S**tore Pressure for Zones 1 to 10'). Ranges for the respective parameter, separated by commas must follow these keywords. '-S' means that the values for the given parameters will be stored in the respective *.CSO file. '-T' will cause the mean values to be written in the output (.COF) file.

The following table gives a list of keywords which are currently available:

PZ	Pressure per Zone
TZ	Air Temperature per Zone
FZ	Flow per Zone
HZ	Humidity per Zone
IZ	Outdoor air infiltration per Zone
AZ	Outdoor air change rate
MZ	Room mean age of air per Zone
EZ	Room air change index per Zone
C*n*	Concentration per Contaminant and Zone, 1<= n <= 5; n = contaminant number
Q*n*	Pollutant Source Strength per Contaminant and Zone, 1<= n <= 5
S*n*	Pollutant Sink per Contaminant and Zone, 1<= n <= 5
FL	Mass Flow per Link (Note: Normally the net flow rate is given. To get the flow in each direction put a "<" in front of the link name for flow *To* to *From* and a ">" in front of the link name for flow *From* to *To*).
TL	Temperature per Link
VL	Actual value per Link
PE	Wind pressure per external node
WA	Wind Velocity
TA	Outdoor Air Temperature
HA	Outdoor Humidity
FB	Flow matrix for the building
IB	Outdoor Air Infiltration for the building
AB	Outdoor Air Change Rate for the building
MB	arithmetic mean of building mean age of air
RB	RMS of building mean age of air
NB	Nominal time constant of building mean age of air
EB	Air Change Efficiency of building
LB	Ventilation Heat Loss Energy of building

To get the flow matrix with mean values, the ‘FB-T’ option has to be specified in addition to ‘FB-S’, because the matrix with the mean values will be written to the *.CSO file and not into the *.COF file.

The definition of `Temperature per Zone' (TZ) as given in the example means that the values for zones 1 to 20 are stored in the TZ.CSO file ('TZ-S' keyword). The 'TZ-T' makes COMVEN write the values in the output file. Note that the definition of `Ambient Temperature' ('TA') given in the example does not have any parameters, since this is an entity assigned to the outdoor conditions as a whole, not dependent on a parameter such as `zone' or `link'. The specification `C1-S 1,2,3' as given in the example means "create output in a .CSO file for concentration of gas number 1 in zones 1, 2 and 3.”

Histograms in COMIS

'Concentrations' for zones and/or occupants can be directed to histograms. The 'Limit Concentration' will be used to flag 'Excess concentration score'

'Flow rates' for zones and/or occupants (flow rate in the zone in which they stay) can be directed to histograms.

'Effective Flow rates' can be calculated in a number of ways for zones and for occupants.

'Effective Flow rates' are calculated from a room concentration. COMIS always uses the concentration of pollutant 1 for the effective flow rate calculation.

In the calculation a 'fictive, constant source strength' is assumed (input at **&-POL-DEScription**). The actual source strength may be time dependent and differ in size, given by the occupant activity or by the schedule and initial value per zone.

EF=Effective flow rate (at timestep t)
FQ=Fictive Source
Af=Floor Area
Aw=Wall Area (all walls including ceiling and floor)
Vz=Volume of the Zone
Ct=Concentration of the zone at a timestep t

For zones there are 4 ways to calculate the Effective flow rate controlled by the parameter 'RoomDep' in &-HISTO:

1) fixed fictive source strength (roomDep=0)

$$EF = \frac{FQ1 \times 1}{Ct}$$

2) fictive source strength floor area dependent (roomDep=1)

$$EF = \frac{FQf \times Af}{Ct}$$

3) fictive source strength wall area dependent (roomDep=2)

$$EF = \frac{FQw \times Aw}{Ct}$$

4) fictive source strength room volume dependent (roomDep=3)

$$EF = \frac{FQv \times Vz}{Ct}$$

For occupants there are also 4 ways to calculate the Effective flow rate controlled parameter 'OccuDep' in **&-HISTO**:

1) fixed fictive source strength (OccuDep=0)

$$EF = \frac{FQo \times 1}{Ct}$$

2) fictive source strength proportional to the number of occupants (OccuDep=1)

$$EF = \frac{FQo \times OccNum}{CT}$$

3) fictive source strength proportional to the occupantactivity (OccuDep=2)

$$EF = \frac{FQo \times OccAct}{CT}$$

4) fictive source strength * activity*number of occupants (OccuDep=3)

$$EF = \frac{FQo \times OccNum \times OccAct}{Ct}$$

The calculation and output of histograms can be opted for at &-PR-SIMU options. Instead of the -S or -T option **-Hnn** must be used. The number nn may be 1..20 and is the type number of the histogram (explained later).

Example
&-PR-SIMU
...
C1-H1 1-3, kit, hoo, zon1, occ3
C2-H1 hal
C3-H12 1,occ1,occ3
FZ-H4 zon1,zon2,occ1
FE-H2 zon1,zon2,occ1

C1-H ... will produce a histogram (type 1) for Concentration 1 (which is for the first pollutant). The histogram is multiple and like a table includes a range of different rooms 1, 2, 3, kit, hoo, zon1 and occ3; each, in their own column. Occupants may be included in the range. The concentration for an occupant is the concentration of the room the occupant happens to be in. This may change during the simulation as occupants may be scheduled to move from room to room.
FZ-H ... stores the Flow per zone in a histogram
FE-H ... stores the effective flow per zone in a histogram
The types of histograms are defined in a new keyword section. If Concentrations are used in the histogram, then this section must come after &-POL-DEScription !

&-HISTO defines ranges for histograms. The ranges follow the User Units delcared before this keyword section. So the lower and upper class must be defined in User Units. Histogram Types can be used for more than one output variable (here H1 is used for C1 but also for C2) that can have individual conversion factors.

Histograms are invoked by:

- C1-H1 zon1, zon2, occ1 <---concentration histogram
- FZ-H4 zon1, zon2, occ1 <---Flow rate histogram
- FE-H2 zon1, zon2, occ1 <---Effective flow rate histogram
- effective flow rate = (Norm Source)/(current Concentration) USER UNITS !
- Norm Conc can be used to signal undesirable situations

Histogram (-)	Number of Classes (-)	Lower Class (-)	Upper Class (-)	Room Size Dependency (-)	Occupant Dependency (-)

Example input:

1	*11*	*0*	*1000*	*3*	*0*
2	*11*	*0*	*100*	*0*	*1*
3	*16*	*0*	*10*	*1*	*2*

COMIS does not have input yet for floor area, so the W L H input at gradients is taken. The actual concentration used for the calculation of the effective flow rate is the concentration of pollutant 1.

Example:
Occupants are used to produce CO_2 as pollutant 1 (dependent on their activity and size) and move from room to room. The effective flow is in a histogram. The Norm Concentration is set to 600E-6 (600 ppm) (as delta concentration above ambient concentration, but we simulate here at zero outside concentration). The absolute CO2 level would then be 1000..1100 ppm.

The fictive source is 5.5E-6 m^3/s CO_2 which is about the average value for a person at light work. The room size dependency is set to 0 (fixed) in this case. The histogram will now contain the effective flow rate per person. If this person entered a room ventilated by infiltration just for one person only in which nobody has been, the concentration will start at 0 leading to an infinite effective ventilation flow rate. The concentration will rise to a steady state close to the Norm Concentration.

The first effective flow rates will fall outside the classes of the histogram and will be counted in the class for overflow values. In fact this histogram shows the interesting part for us where the effective flow rate may become too low.

If the room size dependency would be chosen as 3, then the effective flow rate would be per room, but becomes undefined if the number of occupants in that zone goes to zero. The program maintains a minimum of 1 occupant per room in the case (only for this effective flow calculation, this fictive person is no source in the pollutant transport model).

For sources that are more or less proportional to the floor area the room size dependency is set to 1 and the effective flow rate becomes a value per unit of floor area. (If wall area would be the source, just input that, as COMIS does not 'know' anything of these area inputs). In this situation sources from building materials or furniture can be judged.

A similar possibility is for volume dependent sources.

The effective flow rate is higher than the actual (fresh air) flow rate in a room if the concentration is below the steady state concentration. The effective flow rate is lower than the actual ventilation in the room if the incoming air passes other rooms in which the concentration is already raised. By definition the effective flow rate leads to a concentration that would exist at steady state in a test room with the same source and the same flow rate as fresh air supply. Therefore it is the best value to look at when pollutant problems are to be solved. For energy loss/consumption it is better to look at the infiltration flow rate itself.

5.2.3 Problem Control Parameter Definition

The solver used in COMVEN is the Walton[1] solver with fixed relaxation factors. It makes use of the air flow rate and the derivative d/dP of the flow per air flow component.

Keyword:

&-PR-CONTrol parameters **--- OPTIONAL DATABLOCK ---**

Header:

1.	Tolerances				
Under Relaxation Factor [-]	absolute EpsFA [kg/s]	Relative EpsFR [-]	CORR*JAC(i,i) EpsCJ [kg/s]	Start Number of Iterations [-]	Link Flow Pressure Laminar Flow DifLim [Pa]

Example input:

0.75	*1.E-6*	*1.E-5*	*1.E-7*	*2*	*2.E-5*

Initial Newton iteration steps:

The Solver is a Walton with fixed relaxation factors. This is a very simple solver that only switches the relaxation factor, depending on the convergence, but the performance is very good.

convergence	relaxation factor
<=0.3	1.0
0.3 to 0.7	0.75
> 0.7	0.5

The solver first starts with a number of pure Newton iterations using the under-relaxation factor. This number is given as user input under &-PR-CONT Header 1 data 5 (start number of Newton iterations). Useful values are 2 or 1. After those iterations the Walton solver is used.

[1] The Solver has been developed by George Walton of NIST for the AIRNET program

Description:

Parameters	Description	Default value
Under Relaxation Factor	Newton pressure corrections are multiplied with this factor if pure Newton iteration steps are used. Practical range Relax=0.5,...,1.0	1.0
Tolerances Abs EpsFA Rel EpsFR	Absolute tolerance per flow Relative tolerance per flow (100% is total flow through the zone)	1.0E-6 kg/s
CORR*JAC(i,i)	Newton pressure correction multiplied with the derivative d(flowsum per zone)/d(zone pressure) This is an estimation of the error in the sum of the flow per zone.	1.0E-4
EpsCJ	Tolerance for CORR*JAC(i,i)	0.2*EpsFA
Start no of Iterations	The number of pure Newton iteration steps before switching over to optimum relaxation.	1
DifLim	Link flows are approximated by a linear function at pressures of which the magnitude is smaller than DifLim. This is effective to avoid problems with large openings	1.0E-4 Pa

Header:

2.	use old Pressures	No Pressure Initialization	Max Number of Iterations allowed
	0=Zero Pressures 1=use Previous 2=recalculate air density after every iteration step	0=Lin.initial. 1=No initial.	
	UseOPz [-]	NoInit [-]	Miter [-]

Example input:

1 *0* *500*

Description:		
Parameters	**Description**	**Default value**
Use Old Pressures	UseOPz determines what the program takes for the pressures per zone at every time step. This pressure has some influence on the air density in the zone and thus on the resulting stack effect, especially if the zone reference height is far away from the level of most links, and if the zone pressures are large. UseOPz=0 zeroes the initial pressures per zone per time step. UseOPz=1 uses the pressure from the previous time step as initial pressure. UseOPz=2 recalculates the air density with the pressure from the previous iteration step	0
No Pressure Initialization	The program has the option to start with a first pressure iteration based on a linear network derived from the give C-value	0
Max Number of Iterations allowed	If a solution is not found before the given number of allowed iteration steps, the program breaks and reports that the output for this time step may be wrong	50

Convergence problems in COMIS.

----------------FA, F1 through F4, RF and (other) unstable networks---------------------------

Definition:
The overall resistance from a node is the replacement resistance of all links to the node. FA F1..4 and RF impose a thread that the overall resistance of a node becomes negative.

Definition:
"Negative-Nodes" are nodes for which:

more air flows out of the node
if the pressure is lowered,
less air flows out (or more in) if the pressure is raised.

"Normal nodes", with passive resistances have the reverse:
more air flows out if the node pressure is made higher
more air flows in if the node pressure is made lower

All nodes with an overall negative resistance lead to a situation where the network cannot be solved. The zone pressure cannot converge.

Tests for negative-nodes could be done, but zones can become negative-nodes depending on the pressures in the network. At some pressure distribution the node becomes locally negative. As the solver jumps pretty irregular through the pressures, this problem can occur in some cases in the numerical solution while it would not exist in reality.

On the other hand: negative nodes almost never exist in reality.

The few examples are:
1. an ill-conditioned control loop for a fan with positive feedback (this will lead to vacuum or the maximum pressure the fan can blow into the node).
2. the detonation of an explosive (chemical reaction speeds up as the pressure rises, making the thing unstable)
3. a loose hanging flap/part in a wall closing at a certain pressure.
4. a flow controller! (it does the same as 3)

The message COMIS gives is:

MATRIX SINGULAR in ZONE # nn.

The number nn can be the negative node at either side of the negative-link.

Negative nodes can be caused by negative parts in links with:
1. Fan characteristics (at an ill polynomial).
2. Flow controller characteristics.
3. RF (Related Flow) characteristics.

These three characteristics can be (locally) negative (flow over the component, in the direction of the pressure vector, drops as pressure rises). Depending on the other links and points of operation on their characteristic a negative-node could result.

COMIS has some added network checks which prevent most of the 'MATRIX SINGULAR' conditions. If one gets this message and fans, flow controller or RF components are in the network, one should try first to replace these components step by step with a standard CR component. This will probably locate the cause of the negative node.

For fans negative parts in the curve do not exist. The fan in COMIS is described as Fma = f(dp). This should not be confused with the pressure bump (contraflexure) seen in fans with forward bend scoops. One cannot input such a curve in COMIS!

At flow controllers one should watch for the part where the flow decreases if pressure increases. If this decreasing flow rate is not compensated by an increasing flow rate in other links in the zone in which the flow controller is placed, then the node becomes negative.

RF components that have their reference link connected to the same node, and have a flow direction opposite to the flow of the reference link, and have a larger flow rate than the reference link form a negative link. If this is not compensated by (large enough) other links, the node will get negative.

RF components in zones further away from their reference link may cause a series of nodes to become negative, but only if their flow rate is (much) larger and opposite to the flow in the reference link. If a large exchange flow rate must be provided by a RF component, then one should use two RF links with the same RF type, but link them in opposite directions to the node. This will never cause convergence trouble.

5.4 Network Description

5.4.1 Air Flow Components

Main Keyword:

&-NET-AIR flow components

Header:

```
# Allowed prefixes are:
#*CR   *FA    *DS    *DF    *F1    *F2    *F3    *F4    *WI    *TD    *RF
#  |      |       |       |      |                      |        |        |        |
#  |      |       |       |      |______________________|        |        |     related flow
#  crack |        duct    |        flow-controllers               |     testdata points
#         fan            duct-fitting                          window(openable)
# keep the KEYWORDS   &-CR,...,&-TD in this part &-NET-AIR
```

Caution: Legal length of air flow component name including prefix is 10 characters!

5.4.1.1 Crack

Keyword:

&-CR crack

Header:

1.	Prefix and Name (-)	Description [-]

Example Input:

**CR1* *Crack Example*

2. Cs (kg/s@1Pa)	Exp n (-)	Length [m]	Wall Properties Thickness [m]	 U-value [W/m2 K]

Example input:

0.003 *0.667*

Filter values (see page 5.4.2) must follow each crack definition.

Description:			
Parameters	Description	Input Format	Default
Cs	Air mass flow coefficient	Real	0.001
Exp n	Air flow exponent	0.5...1.0	
Length	Crack length	Real	1.
Wall Properties	needed for crack temperature correction routines		
Thickness	thickness of the wall	Real	0.0 ‡
U-Value	Heat transfer coefficient	Real	0.3

‡ A wall thickness >0.0 must be entered in order to initiate the temperature correction calculation for air flows through the wall crack (see COMIS Fundamentals Section 2.2.1.2.6.4 "Walls"). Default thickness = 0.0 means the crack temperature will be averaged between the node temperatures on both sides of the link.

The length of the crack used for describing the flow properties need not be the real length unless a wall thickness is entered as above. In that case the real crack length must be given. The Cm value used in the program equals Cs multiplied by the ratio of the actual value in NET-LINKS (column 8) and the given value for length under CR header 2 column 3. If the real length of the crack is given here the actual value of the crack for a certain link in NET-LINKS [Section 5.4.4 (column 8)] must also be the real actual value.

$$Cm = \frac{\text{Actual Value}}{\text{Length}} Cs$$

If no temperature correction has been calculated, it is perfectly satisfactory to give a "length" of 1.0 for a crack that is in fact 100 m and an actual value of 1.0. Only the given value will be used. In a building with only slightly different leakages in the various facades or windows, giving values of 1.1, 0.95, 1.3, and so on or the actual value can provide a very realistic simulation while only one (average or normalized) crack is in the section AFC, NET-AIR flow components.

3.	Filter 1 (-)	Filter 2 [-]	Filter 3 [-]	Filter 4 [-]	Filter 5 [-]

Example input:

0.0 0.3 0.7

The values given for filters represent the "filter efficiency"; this means the fractions of the (maximum) five pollutants that will not pass through the link. Filter = 0.3 means 30% disappear in the link, 70% go through.

NOTE: At least one filter value must appear for EACH crack.

The input for a set of cracks would look like this:

**CR1 Window_cracks*
0.01 0.667
0.0
**CR2 Door_cracks*
0.03 0.60
0.0
**CR3 Wall_crack*
0.005 0.85
0.0 0.5 0.9

5.4.1.1.1 Standard Conditions for Crack Data

Keyword:

&-NORMCR Standard condition for crack data

Header:

Standard Temperature for the Crack Data (default = 20oC) (^{o}C)	Standard Barometric Pressure for the Crack Data (default 101.325 kPa) [kPa]	Standard Humidity for Crack Data (default 0 g/kg) [g/kg]

Example input:

18 101.320 5

The conditions must reflect the situation during the measurements of the provided crack data or the conditions to which the data has been normalized. Compared to the default temperature, the lower normalized value will provide lower mass flow rates in the network.

5.4.1.2 Fan

Keyword:

&-FA fan

Header:
line1= Fan name.... #line2=flag.... #line3=Pminimum.... #line4=CO.....
line4 up to maybe line8=data pairs
the last line is always the filter line

1.	Prefix and Name (-)	Description [-]

Example input:
FA1 *Bathroom Exhaust Fan*
or
FA-EX3 *(Coefficients are given to describe fan-curve)*

2.	# Flag:	1 = Polynomial input provided by **user** 2= Polynomial input calculated by **interface** from data pairs 3= Polynomial input calculated by **COMIS** from data pairs			
Flag (-)	Exp Polynom. (-)	RhoI (kg/m3)	NfI [rpm]	Cs [kg/s@1Pa]	Exp n [-]

Example input:

1	*3*	*1.2*	*1*	*0.00001*	*0.5*
or:					
2	*2*	*1.1*	*1400*	*0.1*	*0.6*

Note: it is always safer to use data pairs (Flag 2 and 3). COMIS 3.0 uses either the polynomial input (Flag 1 and 2) or calculates the polynomial description for the fan (Flag 3)!

Description:		
Parameters	Description	Input Format
Flag	Choice of input	
Exp Polynom	Polynomial used to describe fan curve. Highest power used is the lower of the number of data pairs minus one or number given in column 2, number has to be less than 6!	Real < 6
RhoI	Air density at fan inlet	Real
Nfl	Revolution per minute, Fan speed used to determine fan curve	Real
Cs	Flow coefficient if fan is off	Real
Exp n	Flow exponent for fan off	Real

If fan performance is described by pairs of data on pressure-rise and volume flow-rate, this input must either be processed by the user-interface (Flag = 2) or by COMIS (Flag = 3) to result in a polynomial curve approximation. Setting the flag under header 2 to "3" causes COMIS to look at the data pairs and calculate a polynomial in the pressure range spanned by the input pressures. The highest power of the polynomial is given by the data in column 2 under header 2. The highest power possible is always the smaller of either the number of data pairs reduced by one or the given value in column 2.

As polynomial fan approximations tend to oscillate one needs to check this with a graph of the polynomial.

If the flag is set to 1 COMIS expects input values for Pmin, Pmax, the slope and intercept under header 3 as well as the coefficients C0, through Cn under header 4.

Following the fan laws the flow through the fan is calculated for deviating values of fan speed and air density. (It is possible to input a normalized fan speed of 1.0 under FA header 2 column 4 and have a 10% faster running fan in the NET-LINKS section by setting the multiplication factor (column 8) there to 1.1. Or one can input 1400 rpm under FA header 2 column 4 and 1540 rpm under NET-LINKS column 8.) If the fan is off (actual rpm less than 1% of the test fan speed (Nfl)) the fan will be simulated with a bypass aperture of Cs and ExpN given in FA header 2 column 5 and 6.

Outside the range of the spanned pressure a linear approximation is used to ensure that the network solver will find a valid solution and not become stuck outside the correctly approximated curve. This linear part has also to be calculated by the user-interface and the values for that are under header 3. The pressure range (Pmin-Pmax) will normally run from 0Pa up to the pressure where the flow through the fan is zero, as seen from the polynomial. However if fan-data points are given for reversed flow (which is unusual for fans) or in the quadrant of reversed pressure (another fan in series over-blows this fan so that it forms a resistance rather than a pump) then the polynomial will be used for that spanned range. In this case the linear section starts on both sides where the data ends.

3.	Pmin (Pa)	Pmax (Pa)	Slope (m3/s/Pa)	Intercept (m3/s)

Example input:

	0	*57.8*	*-0.73*	*42*

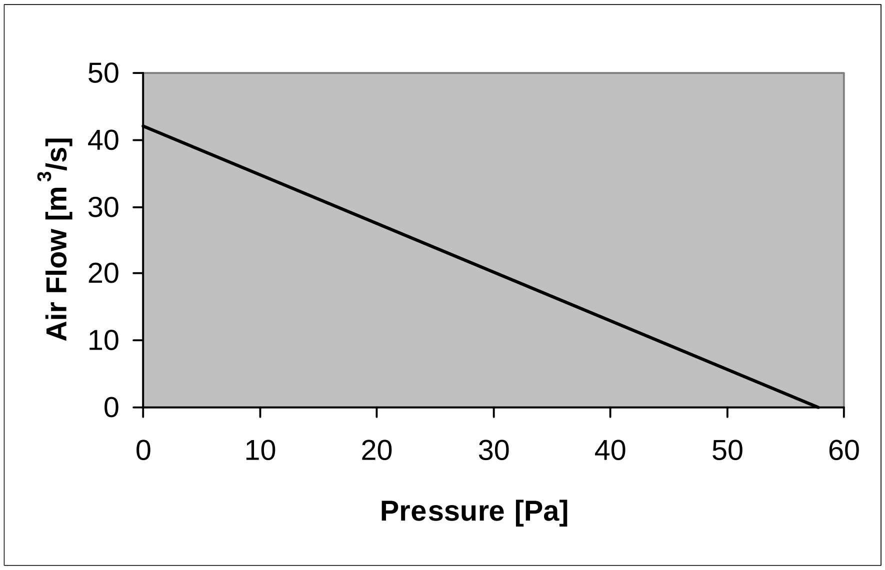

Figure 5.4.1: Example Input for Header 3; (Slope is range of air flow divided by range of pressure)

Description:		
Parameters	Description	Input Format
Pmin	lowest pressure point	Real
Pmax	highest pressure point	Real
Slope	slope of the linear extrapolation	Real
Intercept	intercept between the curve and the flow axis	Real

4.	CO (m3/s)	C1 [m3/s/Pa]	C2 [../Pa2]	C3 [../Pa3]	C4 [../Pa4]	C5 [../Pa5]

Example input:

42 *-0.11* *-0.01* *-0.8E-7*

This calculates the fan performance as follows: $Q = 42 - 0.11\Delta p - 0.01\Delta p^2 - 0.8^{-7}\Delta p^3$

Description:		
Parameters	Description	Input Format
C0 ..C5	Coefficients	Real

It should be noticed that the fan curve is described as flow (y) vs. pressure (x)!

5.	Fan Curve: Pressure Rise vs. Flow Rate, maximum 4 Lines Data Pairs: minimum 3 Pairs, maximum 12 Pairs				
(Pa)	(m3/s)	(Pa)	(m3/s)	(Pa)	(m3/s)

Example input:

10	*40*	*20*	*35*	*30*	*30*
40	*20*	*50*	*10*		

Description:		
Parameters	Description	Input Format
Pressure	Pressure Value	Real
Flow rate	Flow rate Value	Real

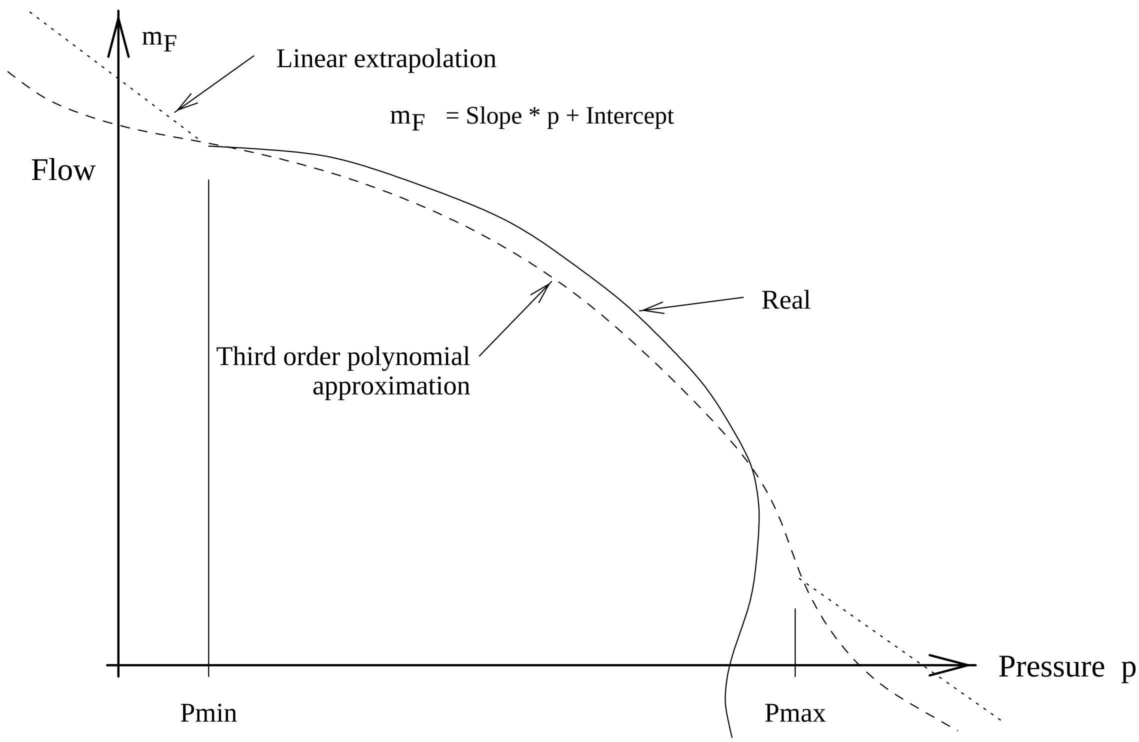

Figure 5.4.2: *Fan Curve Approximation using Coefficients*

9.	Filter 1	Filter 2	Filter 3	Filter 4	Filter 5
	(-)	[-]	[-]	[-]	[-]

For description of filter data, please see Section 5.4.1.1 - Crack

5.4.1.3 Ducts

5.4.1.3.1 Straight Duct

Keyword:

&-DS duct straight

Header:

a duct fitting given here will result in an extra dynamic pressure loss, Zeta

1.	Prefix and Name (-)	Description [-]

Example input:

**DS-1* *Round duct*

2.	Ducts straight Part				one Fitting		
Diam1 (m)	Diam2 (m)	Rough (mm)	Lduct (m)	Zeta [-]	Type [-]	Param1 [acc t]	Param2 [acc t]

Example input

0.3 0.0 0.03 12 1.2

or

0.3 0.3 0.03 12 1.2

Description:

Parameters	Description	Input Format
Diam1	Width of duct or diameter	Real
Diam2	Height of duct	Real
Rough	Roughness of duct	Real
Lduct	Length of straight duct	Real
Zeta	Dynamic loss	Real
Type	see Section 5.4.1.3.2	
Param1	see Section 5.4.1.3.2	
Param2	see Section 5.4.1.3.2	

The duct is characterized by the values of the duct diameters Diam1 and Diam2. If both have a value, a duct with a rectangular cross section is assumed. If only Diam1 has a value and Diam2 is 0.0, a round duct with a diameter of Diam1 is assumed. The wall roughness is given in mm. The length of the duct, Lduct, must be the real length. Zeta represents known dynamic losses. Standard one fitting can be given directly to this duct, and internally, the dynamic loss coefficient of this fitting will be added to the one given under Zeta. As COMIS uses an approximation of the full Colebrook-duct-function, no estimated flow needs to be given.

3.	Filter 1	Filter 2	Filter 3	Filter 4	Filter 5
	(-)	[-]	[-]	[-]	[-]

For description of filter data, please see Section 5.4.1.1 - Crack.

5.4.1.3.2 Duct Fittings

Keyword:

&-DF DUCT Fittings for ducts

Header:

#	Type	Name	No of Param	Parameter	Description param 1 param2
#	1	Entry round[2]	2	t/D	L/D
#	2	Entry round with screen	1	Screen %	-
#	3	Hood	2	Type	Angle
#		Round:		1	
#		Rectangular		2	
#	4	Exit round	-	-	-
#	5	Exit round with screen	1	Screen%	-
#	6	Elbow	1	r/D	-
#	7	Diffuser round	2	A1/A2	Angle
#	8	Contraction round	2	A1/A2	Angle
#	9	Obstruction round duct			
#		Screen	1	screen%	-
#	10	Perforated plate	2	T/DP	N*DP**2/DD**2
#	11	Orifice A†	1	A1/A2	-
#	12	DIN orifice‡	2	A1/A2	L
#	13	Damper	1	Angle	-

T: Thickness of perforated plate [m]
DP: Diameter of hole [m]
N: Number of holes [-]
DD: Diameter of duct [m]

† Routine according COMIS Fundamentals, Ref. 64
‡ This duct fitting is not available yet.

All fittings can only be used together with DS-type components.

Header:

1.	Prefix and Name (-)	Definition [-]

Example Input:

**DF-1* *Diffuser example*

[2] For details see Figure 5.4.3

2.	Type	Param1	Param2
	[-]	[according type]	[according type]

Example input:

7 *30* *0.5*

Duct fittings are separate components of the ductwork. Most fittings can be characterized by two parameters. The actual diameter will not be provided as that has no influence on the dynamic pressure loss coefficient.

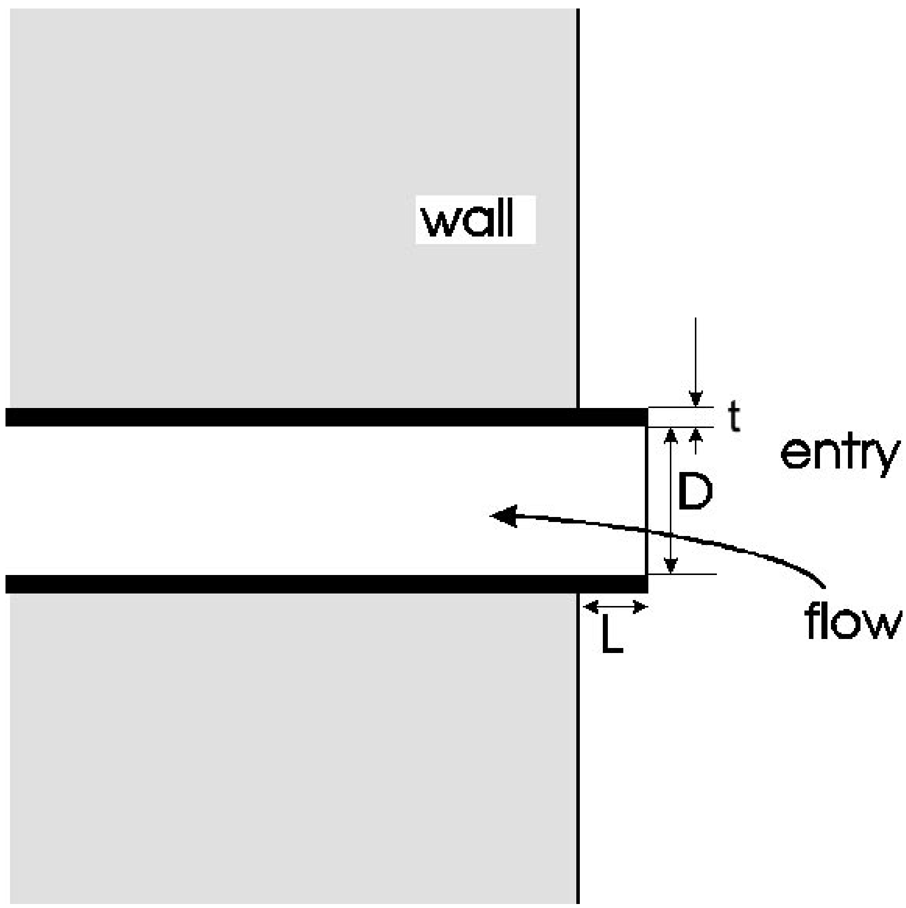

Figure 5.4.3: *Duct Fittings, Type 1: Entry Round*

5.4.1.3.3 Passive Stack

The passive stack AFC component has been made according to a description from **IEA-ECB Annex 27** “Residential Ventilation Systems”[3]

The passive stack is a link from a zone in a building to outside (roof) and has the following elements:

- a grille or opening, visible in the room, mounted on the duct. The opening is a grille (m2) and leads to a pressure loss.
- a duct, which is round if one inputs one parameter, or it is rectangular if one inputs two parameters. The length of the duct (m) is input. The length can be adjusted by giving an **Act. factor** at **&-NET-LINKs.** The friction loss factor (•) is input (if a negative value is provided, a future version of COMIS will interpret this input as the wall roughness needed to calculate the Reynolds-dependent calculation of Lambda).
- Extra pressure losses (zeta's) in the duct can be provided by decreasing the inlet grille accordingly.
- A cowl (hood) at the top of the duct outside. This cowl has a pressure loss coefficient (zeta) for the flow between the duct and outside. The extra Cp-value for the cowl (Cp = 0 ...-1) is added to the outside pressure. At the moment, the meteo wind velocity is being taken as a basis for this calculation.

Annex 27 defined the equation for the passive stack duct :

$$\Delta p = \frac{\varsigma}{2}\left\{\left[\frac{q_v}{A_{grille}}\right]^2 + \left[\frac{q_v}{A_{duct}}\right]^2\left[\frac{\lambda L}{D} + \xi\right]\right\} \ldots + \frac{q_v}{q1}\sqrt{\left[\frac{\varsigma_{out}}{2} v_{meteo}{}^2\left(-Cp_{cowl}\right)\right]}$$

The last term is an extra pressure loss due to the fact that the lower pressure created by the cowl's outside is made more positive by the out-flowing ventilation air. “q1” is the flow through the cowl at 1Pa:

$$q1 = A_{duct}\sqrt{\left(\frac{2}{\varsigma\xi}\right)}$$

Keyword:

&-PS Passive Stack and a Cowl to be mounted from inside to outside

Header:

1.	Prefix and Name	Description
	(-)	[-]

Example input:

*PSround round duct

[3] IEA-ECB: International Energy Agency, Energy Conservation in Buildings and Community Systems program

2. Area of grille (m2)	duct diameter (m)	duct diameter (m)	duct length (m)	duct friction Lambda [-]	duct zeta [-]	cowl zeta [-]	cowl Cp [-]

Example input:
0.02 0.2 0.0 10.4 0.05 0.0 2.5 -0.35

3. Filter 1 (-)	Filter 2 [-]	Filter 3 [-]	Filter 4 [-]	Filter 5 [-]

Example input:
0.0

#The next input is for a rectangular duct

1. Prefix and Name (-)	Definition [-]

Example input:
*PSrect rectangular duct

2. Area of grille (m2)	duct diameter (m)	duct diameter (m)	duct length (m)	duct friction Lambda [-]	duct zeta [-]	cowl zeta [-]	cowl Cp [-]

Example input:
0.0125 0.15 0.2 10.4 0.05 0.0 2.5 -0.35

3. Filter 1 (-)	Filter 2 [-]	Filter 3 [-]	Filter 4 [-]	Filter 5 [-]

Example input:
0.0

How the Passive Stack is being integrated into the network is shown in the section **&-NET-LINKs (5.4.4.2)**

5.4.1.4 Flow Controller

5.4.1.4.1 General

Four types of flow controllers are distinguished which, between them, represent most of the available dampers or regulators as long as the input signal comes from the pressure drop or (duct) flow. Controllers with temperatures as input must be simulated with the schedules -- not yet an ideal situation.
The basic premise of the controllers is that they have an opening through which the air flows. At higher pressures a flap or valve may throttle the flow by gradually closing the opening.

Range 1:
At low pressures this opening is fully open and not blocked by either flap or valve. This part is simulated with a normal crack flow equation, a Cq (volume flow) and an exponent.

Range 2:
Above a certain pressure a moving flap or valve will decrease the opening, thus limiting or keeping the flow roughly constant. This part will be simulated, ideally, by a single constant flow rate, or with a curve approximated by a polynomial. The curve might go up or down or both. The transition point between range 1 and range 2 is simply defined in the program by means of the intersection of both curves. This curve or constant flow will be maintained until the cross point with the curve from range 3.

Range 3:
At high pressures the flap or valve may:
-close no further or have leaks
-close faster, reducing the flow at increasing pressure to almost zero.
Range 3 starts at the intersection of the two curves in the ranges 2 and 3 respectively. Range 3 is also simulated with a polynomial. Care should be taken with steeply decreasing flows. If, due to this, the zone pressure rises if less air is supplied to the zone, or the opposite, then the network can no longer be solved. In reality, such flow controllers will probably also oscillate.

The polynomial approximation will be maintained until the maximum pressure Pmax. As the polynomial approximation could oscillate at higher pressures which would lead to solver problem, a linear approximation above Pmax is taken:
slope = Fva(Pmax)/Pmax and intersept = 0.
If the flow in range 3 is decreasing, the program will look for a root near Pmax in the curve of range 3. Make sure that Pmax is a rough estimation of this root. Above this root the flow will remain zero.

To evaluate the exact pressures at the intersections of the different curves, the program needs a rough estimation of these values, because the curves could in fact have several points of intersection.

The resulting flow is temperature compensated according the relation of air densities at actual and test conditions.

Note: The default units of the parameters for range 2 and 3 are volume flow units. With the unit „fan" in the &-PR-UNIT section they can be changed to mass flow units.

5.4.1.4.2 Flow-Controller, Ideal-Symmetric

Keyword:
&-F1 FLOW CONTROLLER IDEAL SYMMETRIC

Header:

1.	Prefix and Name	Description
	(-)	[-]

Example input:
**F1-EX1* *Flow Controller Ideal Symmetric*

2.	Range 1		Range 2
RhoI (kg/m3)	Cq (m3/s/@1Pa)	Exp n (-)	Fva Setpoint (m3/s)

Example input:
1.2 *0.1* *0.65* *0.2*

Description:

Parameters	Description	Input Format	Default
RhoI	Air density at test conditions	Real	1.2
	Range 1:		
Cq	Flow coefficient of flow controller (volume flow)	Real	0.1
Exp n	Flow exponent of flow controller	Real	0.65
	Range 2:		
Fva_Setpoint	Volume flow at setpoint	Real	0.0

3. Façade Leakage	
Cs (kg/s/@1Pa)	Exp n (-)

Example input:
0.1 *0.65*

Description:			
Parameters	Description	Input Format	Default
Cs	Flow coefficient of facade leakage (mass flow)	Real	0.001
ExpN	Flow exponent of facade leakage	Real	0.65

3.	Filter 1	Filter 2	Filter 3	Filter 4	Filter 5
	(-)	[-]	[-]	[-]	[-]

For description of filter data, please see Section 5.4.1.1 - Crack

With the component F1 a symmetrical ideal flow controller can be modeled. By "symmetrical" it is meant that the curve will be the same for flows in either direction except for the opposite sign of pressure and flow. The word "ideal" means that the controller, when used in a facade, has been compensated for the leak of the facade. In reality this means that a blower door test has given values for the air leaks and the flow controller has a chance of adjustment to reduce its flow curve as a function of pressure. The total effect will be that the sum of the flow through the facade and the flow controller is kept constant until the leak flow through the facade exceeds the set-point flow rate. In this case the flow controller is closed and the total flow will increase with increasing pressure according the leakage characteristic.

The component F1 represents the sum of the flow through the facade and the flow controller. That means no additional link for the leakage may be defined in parallel to F1. With this approach the leakage is assumed to be at the same height as the flow controller. In reality this is mostly not true. That's why we have in reality (but not in the model) a small deviation from the ideal constant flow due to different stack pressures.

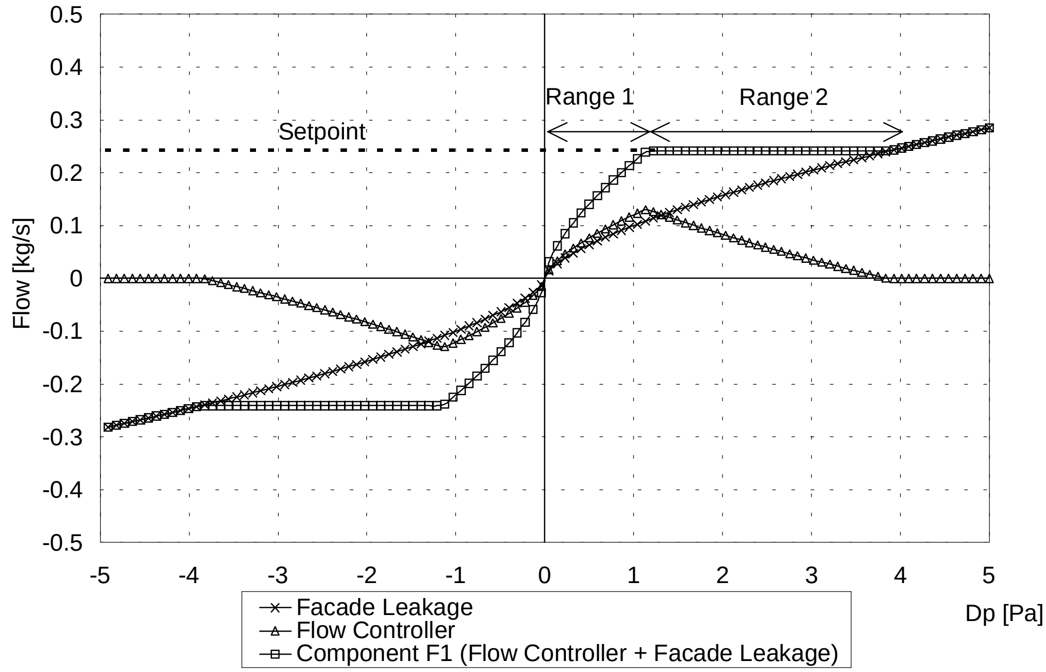

Figure 5.4.4*:* *Component F1 with parameters given in the example input*

5.4.1.4.3 Flow-Controller, Ideal-Non-Symmetric

Keyword

&-F2 FLOW CONTROLLER IDEAL NON SYMMETRIC

Header

1.	Prefix and Name	Description
	(-)	[-]

Example input:

**F2-EX1 Flow Controller Ideal Non-Symmetric*

1.		Range 1		Range 2 Fva_Setpoint	
	RhoI (kg/m3)	Cq (m3/s@1Pa)	Exp n (-)	positive (m3/s)	negative (m3/s)

Example input

1.2	*0.1*	*0.65*	*0.2*	*-0.15*

Description:			
Parameters	Description	Input Format	Default
RhoI	Air density at test conditions	Real	1.2
Range 1:			
Cq	Flow coefficient of flow controller (volume flow)	Real	0.1
Exp n	Flow exponent of flow controller	Real	0.65
Range 2: Fva_Setpoint			
Positive	Setpoint for positive flow	Real	0.0
Negative	Setpoint for negative flow	Real	0.0

2.	Façade leakage	
	Cs (kg/s @1Pa)	Exp n (-)

Example input:

0.1 *0.65*

Description:			
Parameters	Description	Input Format	Default
Cs	Flow coefficient of facade leakage	Real	0.001
ExpN	Flow exponent of facade leakage	Real	0.65

3.	Filter 1	Filter 2	Filter 3	Filter 4	Filter 5
	(-)	[-]	[-]	[-]	[-]

For description of filter data, please see Section 5.4.1.1 - Crack

F2 is an asymmetric ideal flow controller which is almost identical to F1 but which has a separate flow rate set-point for negative flow-directions. A flow controller F2, placed in a facade, must have its positive flow direction "From" outside and "To" inside. It makes a difference in the result, if the "From" and "To" zones are exchanged under NET-LINKS.

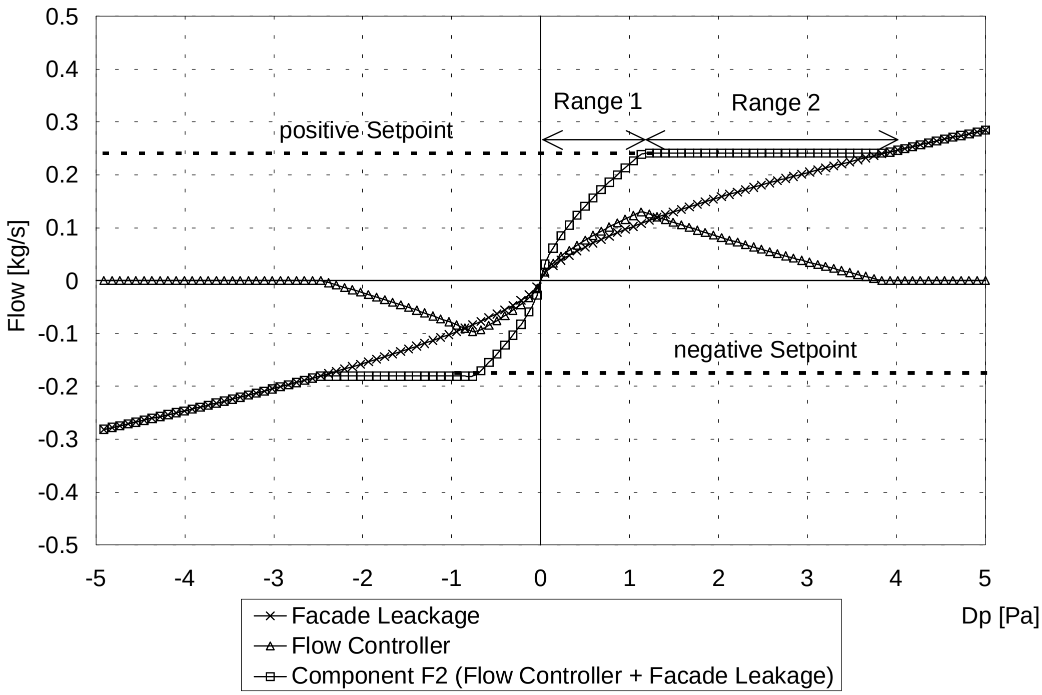

Figure 5.4.5: *Component F2 with parameters given in the example input*

5.4.1.4.4 Flow-Controller, Symmetric, Non-Ideal

Keyword

&-F3 FLOW CONTROLLER, SYMMETRIC, NON-IDEAL

Header

1.	Prefix and Name	Description
	(-)	[-]

Example input:
**F3-EX1 Flow Controller, Symmetric, Non-Ideal*

2.		Range 1	
	RhoI (kg/m3)	Cq (m3/s @1Pa)	Exp n (-)

Example input:
1.2 4. 0.6309298

Description:			
Parameters	Description	Input Format	Default
RhoI	Air density at test conditions	Real	1.2
Range 1:			
Cq	Flow coefficient of flow controller (volume flow)	Real	0.1
Exp n	Flow exponent of flow controller	0.5...1.0	0.65

3.	Range 2					
R2Dp1 (Pa)	R2C0 (m3/s)	R2C1 (m3/s/Pa)	R2C2 (m3/s/Pa2)	R2C3 (m3/s/Pa3)	R2C4 (m3/s/Pa4)	R2C5 (m3/s/Pa5)

Example input:
4.2 5.363935 1.413954 -7.7863522e-2 0. 0. 0.

Description:			
Parameters	Description	Input Format	Default
R2Dp1	Estimation for pressure where range 2 starts	Real	0.0
R2C0… R2C5	Polynomial coefficients for range 2 (volume flow)	Real	0.0

4.	Range 3						
R3Dp1 (Pa)	R3C0 (m3/s)	R3C1 (m3/s/Pa)	R3C2 (m3/s/Pa2)	R3C3 (m3/s/Pa3)	R3C4 (m3/s/Pa4)	R3C5 (m3/s/Pa5)	Pmax (Pa)

Example input:
10.5 0.785716 4.413735 -0.3206652 0. 0. 0.
14.

Description:			
Parameters	Description	Input Format	Default
R3Dp1	Estimation for pressure where range 3 starts	Real	0.0
R3C0...	Polynomial coefficients for	Real	0.0
R3C5	range 3 (volume flow)		
Pmax	Maximum pressure for polynomial approximation in range 3. Above this pressure the flow is proportional to the pressure or zero	Real	0.0

5.	Filter 1	Filter 2	Filter 3	Filter 4	Filter 5

(-)	[-]	[-]	[-]	[-]

For description of filter data, please see Section 5.4.1.1 - Crack

F3 is a symmetric non-ideal flow controller. Non-ideal means that no compensation for other facade leaks is made by the program. Only the actual characteristic of the controller is considered. The ranges 2 and 3 have to be input with the coefficients of their polynomial approximation.

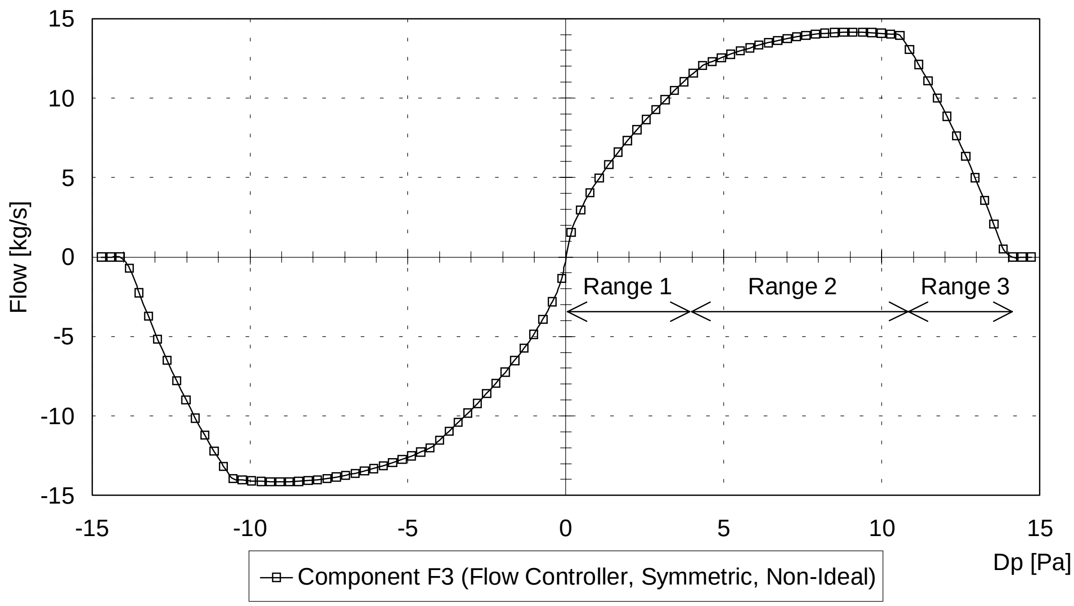

Figure 5.4.6: *Component F3 with parameters given in the example input with decreasing range 3*

5.4.1.4.5 Flow-Controller, Asymmetric, Non-Ideal

Keyword

&-F4 FLOW CONTROLLER, ASYMMETRIC, NON-IDEAL

Header:

1.	Prefix and Name	Description
	(-)	[-]

Example input:
**F4-EX1* *Flow Controller, Asymmetric, Non-Ideal*

2.	Range 1	
RhoI (kg/m3)	Cq (m3/s/1Pa)	Exp n (-)

Example input:

1.2 4. 0.6309298

3.	Range 2 positive flow					
R2Dp1 (Pa)	R2C0 (m3/s)	R2C1 (m3/s/Pa)	R2C2 (m3/s/Pa2)	R2C3 (m3/s/Pa3)	R2C4 (m3/s/Pa4)	R2C5 (m3/s/Pa5)

Example input:

4.2 5.363935 1.413954 -7.7863522E-2 0 0 0

For description, please see range 2 of F3

4.	Range 3 positive flow						
R3Dp1 (Pa)	R3C0 (m3/s)	R3C1 (m3/s/Pa)	R3C2 (m3/s/Pa2)	R3C3 (m3/s/Pa3)	R3C4 (m3/s/Pa4)	R3C5 (m3/s/Pa5)	Pmax (Pa)

Example input:

10.5 111. -20. 1. 0. 0. 0. 13.

For description, please see range 3 of F3

5.	Range 2 negative flow					
R2Dp1 (Pa)	R2C0 (m3/s)	R2C1 (m3/s/Pa)	R2C2 (m3/s/Pa2)	R2C3 (m3/s/Pa3)	R2C4 (m3/s/Pa4)	R2C5 (m3/s/Pa5)

Example input:

-3.5 -51.413954 0.07786325 -0.008 0.0005 0.0001 0

For description, please see range 2 of F3

6.	Range 3 positive flow						
R3Dp1 (Pa)	R3C0 (m3/s)	R3C1 (m3/s/Pa)	R3C2 (m3/s/Pa2)	R3C3 (m3/s/Pa3)	R3C4 (m3/s/Pa4)	R3C5 (m3/s/Pa5)	Pmaxn (Pa)

Example input:

-11. -338. -63. -3. 0. 0. 0. -13

For description, please see range 3 of F3

7.	Filter 1	Filter 2	Filter 3	Filter 4	Filter 5
	(-)	[-]	[-]	[-]	[-]

For description of filter data, please see Section 5.4.1.1 - Crack

F4 is an asymmetric non-ideal flow controller. Parameters for range 2 and 3 have to be defined for positive and negative flow directions respectively. A flow controller F4, placed in a facade, must have its positive flow direction "From" outside and "To" inside. It makes a difference in the result, if the "From" and "To" zones are exchanged under NET-LINKS.

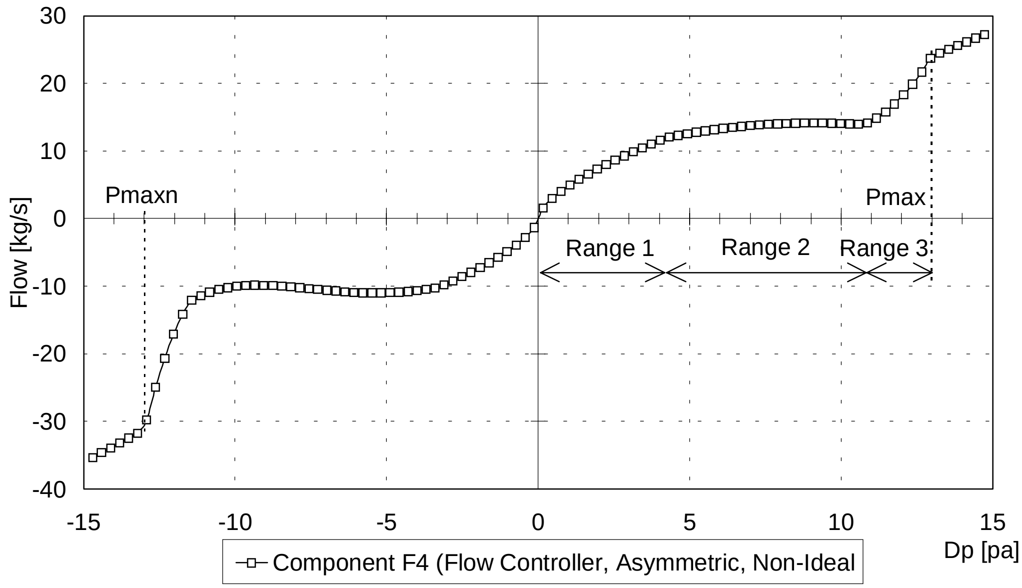

Figure 5.4.7: *Component F4 with parameters given in the example input with increasing range 3*

5.4.1.5 Windows and Doors

A closed window or door can be simulated either with a simple crack CR or with the air flow component WI (WIndow). The difference between these two possibilities is:

- The crack of a CR component is located at the link height given in &-NET-LINKS.
- The crack of a closed WI component is distributed over the height of the window (Lhmax) as follows:
  - bottom crack located at link height (crack length=Lwmax)
  - top crack located at link height plus Lhmax (crack length=Lwmax)
  - plus "integration" over a crack which is vertically distributed between link height and link height plus Lhmax. (crack length=(2 * Lhmax + Lextra))

Due to this distribution over the window height it is possible to have a two way flow through the cracks of a closed window.

A window that is open or will be opened during a simulation run with COMIS can be defined by the air flow component WI.

WI allows the definition of closed windows and doors. In header 2 the cracks and their size are read into the program.

With “LVO Type” (which could be a window or a door) the type of the **L**arge **V**ertical **O**pening can be defined:

- type 1 = rectangular LVO;
- type 2 = LVO with horizontal pivoting axis.

Lwmax is the width of the window and Lhmax is the height. LVOs of type 1 that consist of more than one movable part might have an extra crack length (Lextra). For LVOs type 2 the height of the pivoting axis (Axis height) measured from the bottom of the window frame, can be given. If the window is opened, the size of the opening will be related to Lwmax and Lhmax.

Under header 3 a variety of parameters can be used to give an accurate description of the window opening. The parameter in column 1 is the opening factor: 0.0=closed; 1.0=fully open. For LVO type 2 this range of the opening factor corresponds to the range of opening angle 0 to 90° (example: opening factor 0.5 means opening angle is 45°). At least one line with opening factor 0.0 and one line with 1.0 must be supplied. There may be additional lines between the two. In the simulation the actual value for this window will be read first from the Actual Value in &-NET-LINKS and then from WIndow schedule.

All values between 0.0 and 1.0 indicate that the parameters following on that line are for a partly opened window. During the simulation an interpolation will be made to obtain the best parameters.

In column 2 the discharge coefficient Cd can be given for the opening factor of that line. If the user inputs Cd = 0, the program will determine the Cd from the other input parameters:

Internal doors:	Cd = f(Height of the door/Height of the room); With this function Cd will be in the range from 0.05 to 0.5.
External openings:	Cd = f(wind velocity, temp. difference, depth of the room); With this function Cd varies between 0.6 and 1.5.

If Cd has to be calculated by the program, the following restrictions apply for the input:

- LVO must be type 1
- Cd = 0; for all opening factors

Internal doors:

- The doorstep must be lower than 0.05m
- The zone volume must be input with the H/D/W option. H has to be the same in the "from" and the "to" zone.
- The zone reference height has to be at floor level and the same in the "from" and the "to" zone.

External openings:

- Link to external node (not to a specified pressure)
- The zone volume has to be input with the H/D/W option.
- Only one opened opening per zone is allowed.

The geometry of a partly opened horizontal pivoting axis window (type 2) is determined only by its opening factor. For rectangular openings (type 1) there must be some additional relation factors given (in columns 3, 4 and 5), which define the size and position of the opening in relation to Lwmax and Lhmax:

Opening width = Lwmax * Width Factor
Opening height = Lhmax * Height Factor
Start height = Lhmax * Start Height Factor

Start Height is the distance between the bottom of the window frame and the bottom of the actual opened part of the window (see **Figure 5.4.8**). The sum of Height Factor and Start Height Factor must be less than or equal 1.0 in order to have the actual opening within the window frame.

For slanted windows (e.g. windows in a roof) the Own Height Factor, which is the cosine of the angle between the opening plane and a vertical plane, must be given in &-NET-LINKS. With this factor the program will calculate the start height and the height of the opening in order to get the correct stack pressure difference profile. For calculating the area of the opening, the Own Height Factor will not be taken into account because the flow is assumed to be perpendicular to the opening plane. With this definition it is even possible to input an opening which is horizontal (Own Height Factor = 0). The pressure difference profile is then constant, thus only one-way flow is possible. This is, of course not a correct modeling of a horizontal large opening, which still has to be integrated into COMIS.

Keyword:

&-WI window / door

Header:

1.	Prefix and Name (-)	Description [-]

Example Input:

**WI1* *Window Example*

2.	Closed: Cs (kg/s m @1Pa)	Exp n (-)	LVO Type 1=rectang 2=horizontal pivoting axis (-)	Lwmax (m)	Lhmax (m)	Type1: Lextra Type2: Axisheight (m)

Example input:

0.002	*0.57*	*1*	*2.2*	*1.2*	*1.2*

Description:

Parameters	Description	Input Format	Default
Cs	Flow coefficient for closed windows per m crack length	Real	-
Exp n	Flow exponent	Real	-
LVO Type	Type of Large Vertical Opening 1 = Rectangular LVO 2 = Horizontal Pivoting Axis LVO	Integer	-
Lwmax	Width of Window	Real	-
Lhmax	Height of Window	Real	-
Lextra	Extra crack length for LVOs type 1 with multiple openable parts	Real	0.0
Axisheight	height of the pivoting axis (measured from the bottom of the window frame) for LVOs type 2	Real	0.0

3.	Opening Factor (-)	Cd [-]	Width Factor [-]	Height Factor [-]	Start Height Factor [-]

Example input: (two or more lines are accepted)

0.0	*0.6*	*0.0*	*1.0*	*0.0*
1.0	*0.6*	*1.0*	*1.0*	*0.0*

This table is used for interpolation; the actual value has to be given in &-NET-LINKS!

Description:			
Parameters	Description	Input Format	Default
Opening Factor	Opening factor of window For LVO Type 2: Opening Factor corresponds to the opening angle. (example: Opening Factor 0.5 means opening angle is 45°)	0.0 ... 1.0	-
Cd	Discharge coefficient	Real	0.0
The following are only used for LVO Type 1:			
Width Factor	Actual Lw/Lwmax	0.0 ... 1.0	1.0
Height Factor	Actual Lh/Lhmax	0.0 ... 1.0	1.0
Start Height Factor	Actual Start Height/Lhmax	0.0 ... 1.0	0.0

The absolute FROM or TO height is the total of zone reference plane height plus link height (see **Figure 5.4.8**).

4.	Filter 1 (-)	Filter 2 [-]	Filter 3 [-]	Filter 4 [-]	Filter 5 [-]

For description of filter data, please see Section 5.4.1.1 - Crack

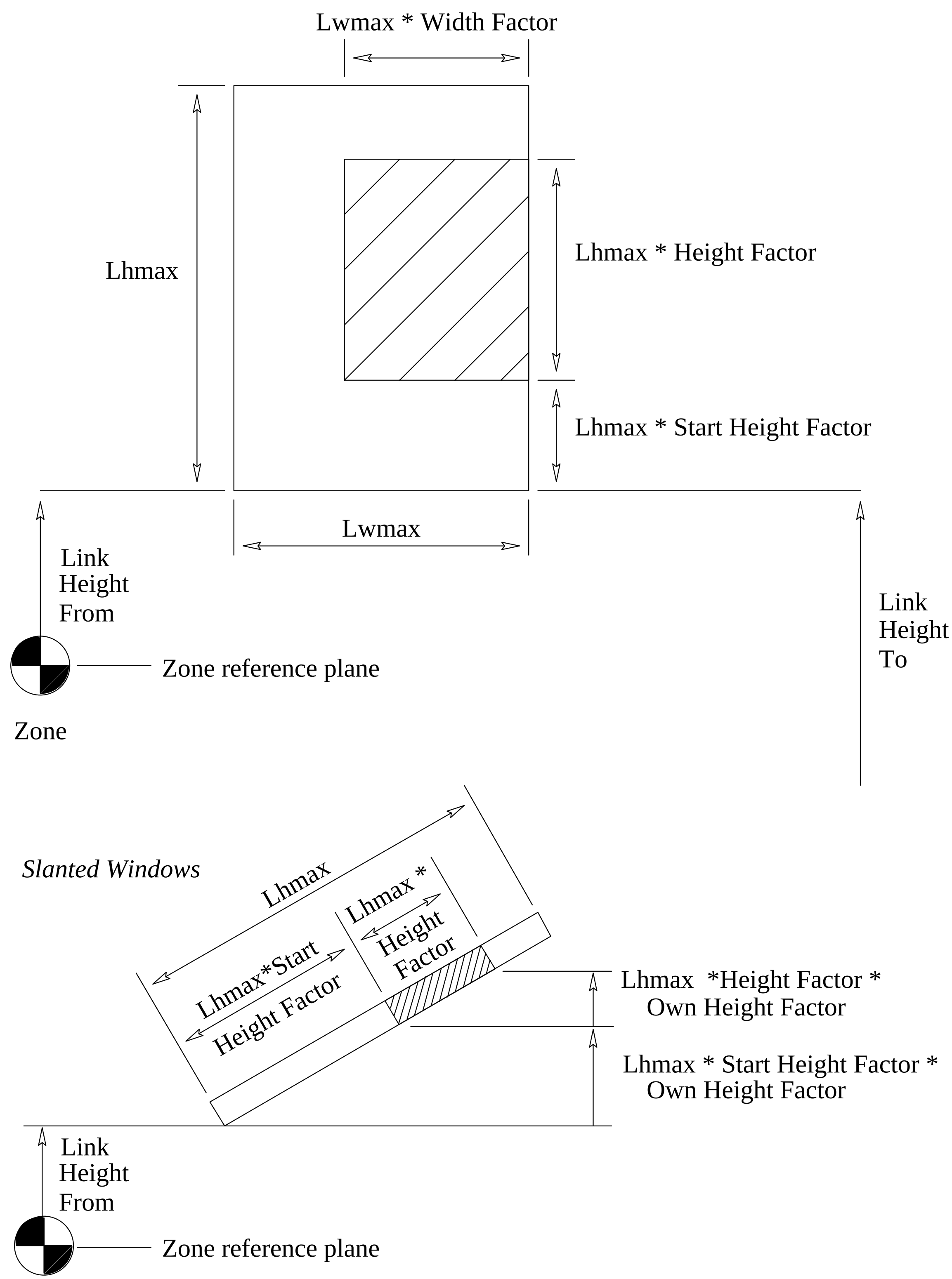

Figure 5.4.8: *Characteristic Measurements for Windows and Doors*

5.4.1.6 Test Data

The Test Data component is there so that measured results can be used in the program COMIS without having to define a separate routine for this. Care should be taken as non-monotone ascending functions might destroy convergence. The input values are given in the same units as the data for the fan. The default is volume flow, however, it can be changed to mass flow by changing the units for the fan flow in **&-PR-UNITS.**

RhoI is the density at which the volume flow rates were assumed; comparable to the fan curve. If one data pair is given a horizontal function is assumed, giving a constant flow at all pressures. If more than 2 pairs are given linear interpolations are made within the data pairs and a linear extrapolation is made outside the maximum and minimum data. It is not yet clear whether the routine will convert all data in a LOG-LOG scale keeping the lowest pressures (below DiFLim) as a linear part.

Keyword:

&-TD test data component(log-log inter-extrapolation)

Header:

1.	Prefix and Name	Description
	(-)	[-]

Example Input:

**TD-1 Testdata*

2.	RhoI
	[kg/m3]

Example input:

1.2

Description:		
Parameters	Description	Input Format
RhoI	Density of air at entry point	Real

3.	Pressure and Flowrate, Maximum 6 lines Data Pairs: minimum 3 Pairs, maximum 18 Pairs				
	(Pa)	(m3/s)	(Pa)	(m3/s)	(Pa)	(m3/s)

Example input:

10.1	*1.0*	*20.0*	*1.5*	*30.3*	*1.9*

4.	Filter 1	Filter 2	Filter 3	Filter 4	Filter 5
	(-)	[-]	[-]	[-]	[-]

For description of filter data, please see Section 5.4.1.1

5.4.1.7 Kitchen Hoods

There are five ways to simulate a kitchen hood in COMIS. **Figure 5.4.9** shows situations called hood 1 - 5.

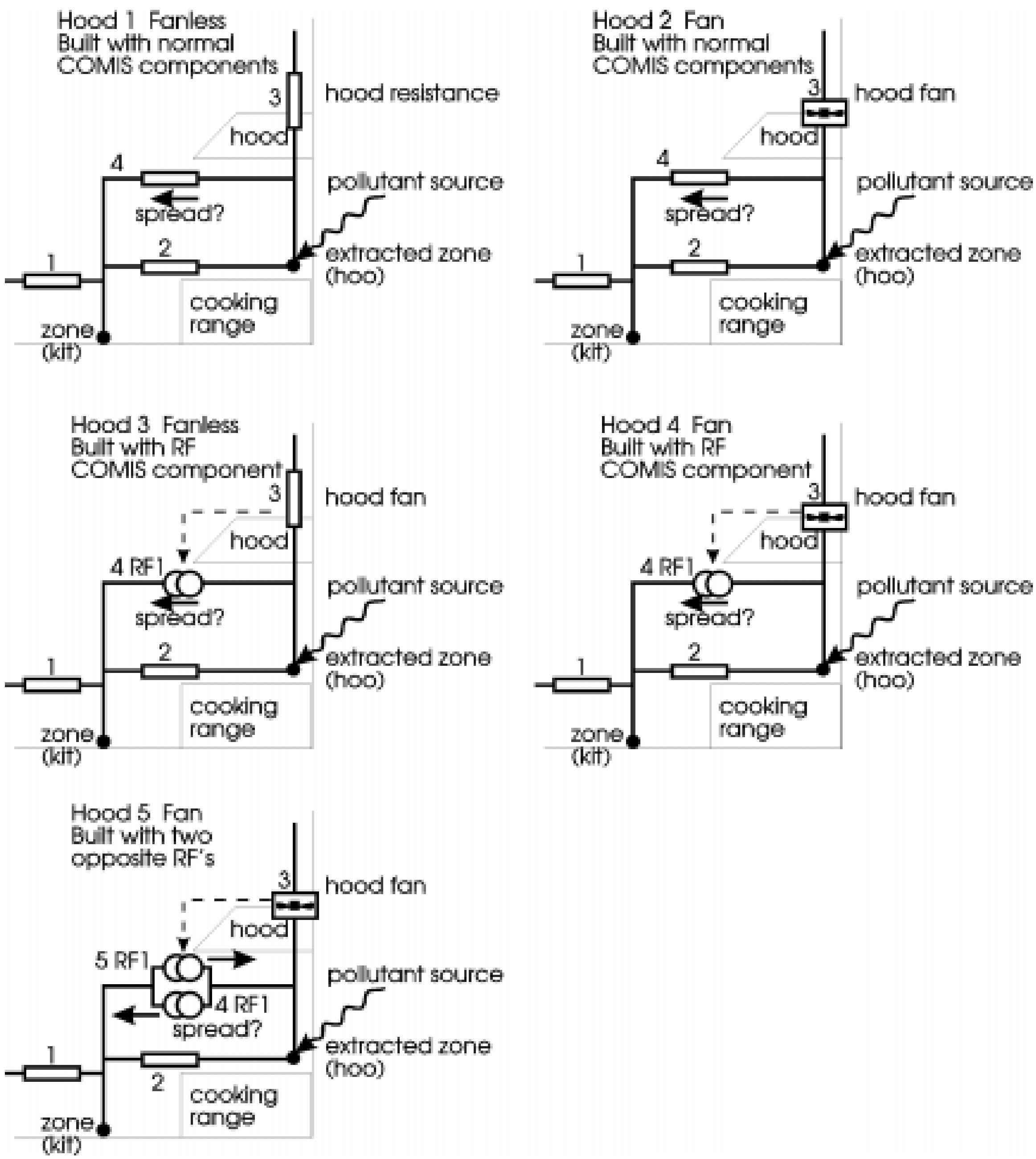

Figure 5.4.9: *Schematics of Fume Hoods as modeled in COMIS*

In Hood 1 link 4 is a normal opening (CR) or a large opening. If the zone above the cooking range is given a higher temperature, spread can occur to the kitchen if the flow through the hood (link 3) is insufficient.

Hood 2 shows the same setup but with a fan in the hood. In both ways the spread is simulated by COMIS, caused by the stack pressure between link 2 and the higher positioned link 4.

If a given hood efficiency is known as a function of the hood extract flowrate (through link 3) then there is a component called RF to model the spread flowrate (link 4). This component RF stands for Related Flow. The flowrate through this component is a function of the flow through a reference link. The function is given in the COMIS input file under AFC at the definition of the RF component used for link 4.

The link to be used as reference link is given in the input file at **&-NET-LINks** in the same column where schedules and reference links for duct junctions are given.

Here follows the part where the RF type is defined

Keyword

&-RF Related Flow COMPONENT

Header:

1.	Prefix and Name	Description
	(-)	[-]

Example input:

**RF1* *Flowhood*

2.	Flag Fva or Fma	RhoI
	(1 or 2)	[kg/m3]

Example input:

2 *1.2*

the 2 above here means the flows given below are volumetric flowrates

3.	Flow rates, Maximum 6 lines Data Pairs: minimum 3 Pairs, maximum 18 Pairs				
(ref m3/a)	(m3/s)	(ref m3/s)	(m3/s)	(ref m3/s)	(m3/s)

Example input:

 # pair 1 pair 2 pair 3
-0.1 0.2 0.02 0.2 0.045 0
 # pair 4
0.4 0

4.	Filter 1	Filter 2	Filter 3	Filter 4	Filter 5
	(-)	[-]	[-]	[-]	[-]

0.0

For a kitchen hood the reference flows shown above are those through the hood, fan and duct. The flow from the cooking range that is leaking/circulating out of the hood into the kitchen zone is given by this RF type. It is the flow q. The efficiency can be defined as (ref-q)/ref*100% for positive q's and positive ref.

The efficiency for the data pairs presented here range from -9900% to 100%.

pair efficiency
1 -
2 -9900% (flow into the room is much more than the hood extract flow)
3 100%
4 100%

Between pair 2 and pair 4 is the normal operation range of the hood. The pairs 1 and 4 have been added to make well defined extrapolations.

How the Hood is being integrated into the network is shown in the section **&- NET-LINKs**

5.4.1.8 Transition

If ducts have been defined and used in the network, transition points from laminar to transition flow and from transition flow to turbulent flow can be given. In the transition range an interpolation is made between the extended laminar function and the turbulent curve. As it depends on the history of the flow or just on chance whether the flow will still be laminar or turbulent there is no single solution in this range. The COMIS simulation model would begin oscillating if these phenomena were included. Keyword:

&-TRANSITion --- OPTIONAL DATASECTION ---

 # If ducts have been defined, ask for ReLam and ReTurb

Header:

ReLam	ReTurb
(-)	(-)

Description				
Element	Description	Value	Default	Unit
1	ReLam	>0	2300	-
2	ReTurb	>0	3500	-

5.4.2 HVAC-Network (is not being implemented in COMIS 3.0)

Duct elements (duct straight DS and duct fitting DF) as well as a test data components (TD) defined in the air flow components section may be linked in series to a duct in this input data section. Note that up to now series of ducts using the Colebrook approximation can only be defined by keeping a zone between the individual duct elements!!

Keyword:

&-NET-HVAc

Header:

Duct Name (*DUxxxxx)	Component 1 (-)	Comp2 (-)	Comp3 [-]	Comp4 [-]	Comp5 [-]	Comp6 [-]

Example input :

**DU1one1*	*DS1*	*DF1*	*DS2*	*DF2*	
**DU2two2*	*DF5*	*DS3*	*DF3*	*DS4*	*DS4*
	DS4	*DF4*	*DS4*	*DF4*	
	DF4	*DS4*	*DF4*	*DS4*	

Description

Parameter	Description	Input Format
Duct Name	Name of duct Note that DU is the fixed two character prefix for the duct name Name must be preceded by a '*' to allow for multiple line input for a particular duct.	String DUxxxxx
Component<i>	Air flow component short name of duct element nr <i> as defined in the AFC input (see Section 5.4) Note that only straight ducts(DS..) or duct fittings (DF..) may be used More than 6 elements may be given using multiple data lines.	String DF.. or DS.. or

5.4.3 Zones Input

5.4.3.1 Zones

Keyword:

&-NET-ZONes

Header:

Zone ID (-)	Name [-]	Temp [C]	Ref. Height [m]	Vol [m3] H/D/W [m/m/m]	Abs. Hum ‡ [g/kg]	Schedule Name [T./H//]

Example input:

E01	*Kitchen*	*18.1*	*-1.00*	*100.*	*10.0*	*T1/H1*
W02	*Living1*	*20.*	*0.50*	*200.*	*12.0*	
S03	*Workshop*	*10*	*-.50*	*200.0*		
N04	*Bedr.Child*	*19*	*0.0*	*150*	*0.0*	*H3*
N05	*Dining*	*21*	*0.0*	*2.4/3.6/3.3*	*1.5*	

Description:			
Parameters	Description	Input Format	Default
Zone ID	Zone Identification	string =8 Char	-
Zone Name	Name of specific zone	string =15 Char, but no blanks between words	-
Temp	Temperature setting in zone	Real	20.0
Ref. Height	Height of zone reference plane measured from building reference plane If the zone is layered or it is a single-sided ventilated zone or if it has an opening for which Cd has to be calculated, then this zone reference plane must be at the floor level of the room. (for details, please see Fig. 5.4.5)	Real	0.0
Vol	Volume of zone	Real	50.0
H/D/W	Height/Depth/Width of the room. This input must be given where there is a single-sided ventilated room, for zones with layers or when Cd should be calculated.	Real/Real/Real	-
Abs. Hum	Absolute Humidity in zone. Only if humidity is used for calculating air density but not transport mechanism.	Real	0.0
Schedule Name	Names of schedules applicable to specific zone string. The first char of the name characterizes the schedule type: T. : Temperature schedule H. : Humidity schedule S. : Sink schedule Q.-OCC. : Source schedule	string = 25 Char. use "/" to separate schedules; no blanks or commas allowed	-

5.4.3.2 Zone Layers

Definition of sub-zones in a specific zone by horizontal partition of the zone into several layers:

Layers can be defined for every zone and also for just a few zones. There is no limit to the number of layers in one zone as long as the total number of layers does not exceed the dimensions of Laydat (currently 50). One restriction is that the layers must be given per zone, in a rising sequence. Every next layer must have a start height, which is higher than the previous layer in the same zone. The start height can be positive or negative and is measured positive above the zone reference plane.

Layers can start anywhere in the zone including outside the range of the floor and ceiling. (The latter is of no use as there will be no link defined there in fact there is no data that tells where the floor and ceiling are.) Links can be defined anywhere above, on or under the reference plane.

Keyword:

&-NET-ZL zone-layers

Header:

Zone *ID (-)	Start Height (m)	Temp Grad [°C/m]	Hum. Grad Factor [1/m]	Poll. Grad Factor [1/m]	Volume Fract [-]	Source Fract [-]	Sink Fract [-]	Flow to next ly Factor [-]	Flow to Zone Factor [-]

Example Input:

**1*	*-.25*	*2.0*	*0.10*	*0.0*	*.33*	*1*	*.33*	*1.5*	*1.1*
	1.5	*3.0*	*-0.15*	*0.0*	*.67*	*0.*	*.67*	*.9*	*0.3*
**123*	*1.1*	*1.5*	*0.0*	*0.0*	*.67*	*0.*	*.67*	*2.0*	*0.0*

Description:			
Parameters	Description	Input Format	Default
Zone ID	Identification of zone in which layers are defined. The zone must be defined!	String	-
Start Height	Height of the lower boundary of the individual layer in reference to the position of the zone node height	Real	0.0
Temp. Grad	Temperature gradient within specific layer	Real	0.0
Hum.Grad Factor	Humidity gradient relative to the moisture content at reference height.	Real	0.0
Poll Grad Fact	Pollutant gradient relative to the pollutant concentration at reference height. The data refer to the pollutant defined as No 1.	Real	0.0
Volume Fraction	Volume fraction of individual layer in relation to total zone volume		0.0
Source Fraction	Fraction of source output of individual layer in relation to zone source		0.0
Sink Fraction	as SOURCE FRACTION		0.0
Flow To Next Ly Factor	The flow to the next layer relative to the total flow through the zone.	†	0.0
Flow To Zone Factor	The relative flow to the fraction of the zone that has not been split up in layers, the flow being relative to the total flow through the zone. If that fraction of the zone is zero, this parameter will influence only the exchange of concentration from layer to layer.	†	0.0

† These parameters are not being used in COMIS 3.0

5.4.3.3 Zone Pollutants

Keyword:

&-NET-ZP zone-pollutants

Header:

Zone *ID	Pollutant		
	Initial Concentration	Source	Sink
(-)	(kg/kg)	[kg/s]	[kg/s]

Example input:

**001*	*0.00*	*0.00*	*1.005*
	0.00	*1.25*	*0.000*
	0.00	*0.00*	*1.005*

The initial values for pollutant concentrations can be given together with the nominal value of the source strength and sink strength. The first data line refers to the pollutant defined as Pollutant No. 1 in the pollutants definition data section. The second data line refers to Pollutant No. 2, and so on.

Description:

Parameters	Description	Input Format	Default
Zone ID	Identification of a defined zone	String	-
Pollutant Initial Concentration	Concentration of specific pollutant at the beginning of the simulation	Real	0.0
Source	Source strength	Real (or name of source or combination of names ‡)	0.0
Sink	Sink strength		0.0

‡ Not available in COMIS 3.0

5.4.3.4 External Nodes

Definition of nodes which are outside the building. The external nodes must be related to the respective facades of the building (see building input) for wind pressure data link.

Keyword:

&-NET-EXternal-node-data

Header:

External Node No (-)	Facade Element No (-)	Outside Conc Factor [-]

Example input:

1	*3*	*0*
2	*5*	*1*
3	*7*	*1*
4	*9*	*1.1*

Description:

Parameters	Description	Input Format	Default
External Node No	External node no. This number appears with a minus sign in the part NET-LINKS	Integer	-
Facade Element No	Number of related facade element Cp-values are linked to the respective facade element	Integer	-
Outside Concentration Factor	Factor of outside pollutant concentration at specific external node in relation to the outdoor pollutant concentration data	Real	1.0

5.4.3.5 Zone Thermal Properties

For single sided ventilation, the temperature drop in a zone caused by opening an external window can be calculated using a simple thermal model of the room. The thermal properties are mean values of all walls in that room.

In addition to the thermal properties, the zone volume must be input with the H/D/W option in &-NET-ZONES and only one window to an external mode must be defined. The window has to be of the LVO type 1 (rectangular vertical opening). If a window with LVO type 2 (horizontal pivoting axis) is selected, the program calculates the temperature drop as if it is a rectangular opening with open area equal to Lhmax x Lwmax.

As soon as the window to outside is opened, the actual zone temperature will be taken as the initial value for the thermal model. The zone temperature as defined in &-NET-ZONES or in a room temperature schedule will be replaced with the time dependent zone temperature calculated with the thermal model.

Keyword:

&-NET-ZT zone-thermal-properties

Header:

Zone *ID (-)	Conductivity (W/mK)	Density (kg/m3)	Capacity (J/kg/K)	Wall Thickness (m)

Example input:

**001 0.8 1420 1000 0.2*

Description:

Parameters	Description	Input Format	Default
Zone ID	Identification of a defined zone	String	-
Conductivity	Mean value of thermal conductivities of all walls in the zone	Real	-
Density	Mean value of the densities of all walls in the zone	Real	-
Capacity	Mean value of the thermal capacities of all walls in the zone	Real	-
Wall Thickness	Mean value of the thickness of all walls in the zone	Real	-

5.4.4 Link Input

Keyword:

&-NET-LINKS

Header:

Link	Type	Zone ID		Height		Own Height	Factor/ Actual RPM/	3Dflow or	Schedule Name (5 Char.)	
ID	Name	From	To	From	To	Factor	Value	Press	T-Junct. No	Ref. Link Angle
(-)	(-)	(-)	(-)	[m]	[m]	[-]	[-]	[Pa]	[-]	[deg]

Example input:

1	*DS-1*	*1*	*2*	*0*	*0*	*1*	*1*	*0*	*2*	*90*
2	*DS-1*	*2*	*3*	*0*	*0*					
3	*DS-1*	*2*	*4*		*0*		*0*			
L1	*DS-1*	*-1*	*2*	*0*	*0*	*d*	*1*	*0*	*l2*	*90*
L2	*DS-1*	*2*	*3*	*0*	*0*					
L3	*DS-1*	*2*	*4Pa*	*0*	*0*					
L4	*WI-1*	*4*	*5*	*0*	*0*	*1*	*1*	*0*	*W1*	
L5	*FA-1*	*5*	*6*	*0*	*0*	*1*	*1*	*0*	*F1*	

Zone identification can relate to internal zones, external nodes (minus sign), or zones with fixed pressures (pressure level is given in Pascal). The absolute FROM and TO height values normally are equal. If the height values are not equal, the link will have a vertical length (e.g., a ventilation shaft). These heights are the sum of zone reference height plus link height (see **Figure 5.4.10**). The program's internal tolerance for warning about differences in the absolute FROM and TO height of the window component is set to 0.02 m.

Description:			
Parameters	Description	Input Format	Default
Link ID	Number of link	String	-
Type Name	Prefix and name of defined air flow component	string =6 char	-
Zone ID From	From zone identification (zone must be defined!)		
Zone ID To	To zone identification		
Height From	Height of link in reference to "From" zone node (see Fig. 5.4.5)	Real	0.0
Height To	Height of link in reference to "To" zone node (see Fig. 5.4.5)	Real	-
Own Height Factor	Only used for slanted windows (e.g., in a roof). Own height factor is the cosine of the angle between opening plane and vertical plane (see Figure 5.4.8).	Real	1.0
Facor/Act. RPM/ Actual Value	For Crack: see Section 5.4.1.1 For Fan: actual RPM or Factor; has to correspond with input provided in Fan section!!! (see 5.4.1.2 for details) For WI: Opening Factor	Real	1.0
3Dflow PRESS	Additional pressure due to 3D flow pattern considerations	Real	0.0
Ref Link No	Reference link for duct joint angle definition (link must be defined!)	Pos int	-
Ref Link Angle	Angle of duct joint to reference link		
Schedule NAME	Name of schedule applicable to the defined air flow component. The schedule type must correspond to the air flow component type!	String	-

No DF-components allowed as link element. T-junction nodes are defined as TO-nodes for the link which represents the main duct part of the T-junction. Default angle for the T-junction is 90 degrees. COMIS 3.0 only allows 90 degree T-junctions.

If the "Height From" and the "Height To" are different, then the flow path is calculated as having a vertical component.

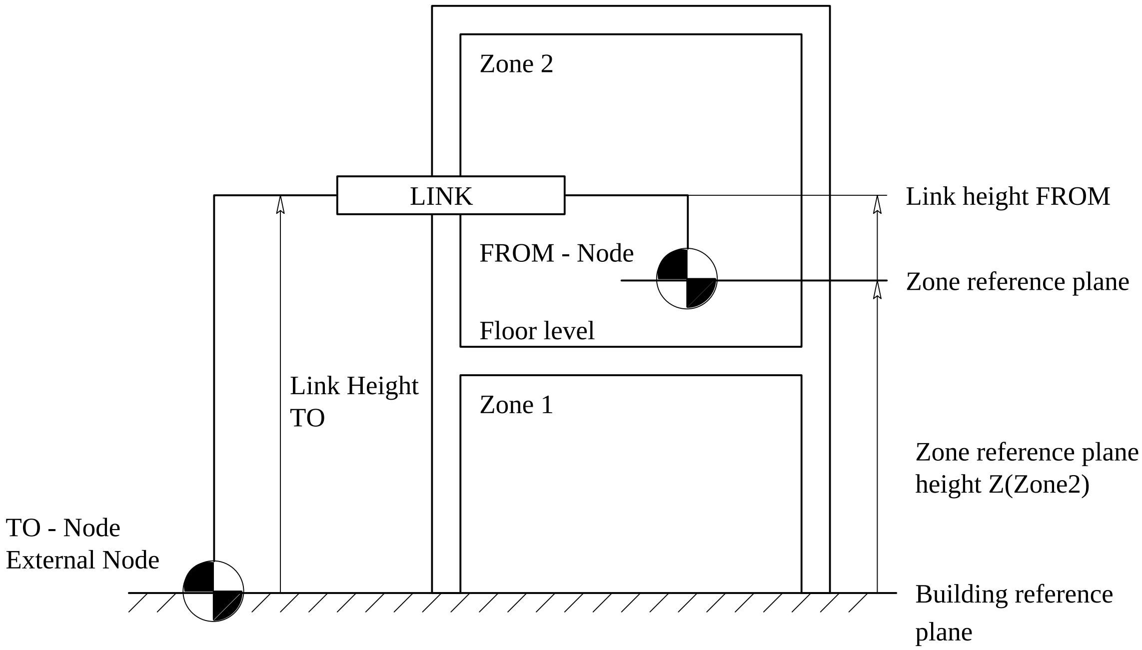

Figure 5.4.10: *Definition of Link Height*

5.4.4.1 Description of Hoods in the Network

Here follows the example of Hood 4 in **&- NET-LINKs**

&-NET-LINks 22

Link	Type	Zone ID		Height		Height	Factor/ Actual RPM/	3Dflow or	Schedule Name (5 Char.)	
									T- Junct.	Ref. Link
No	Name	From	To	From	To	Factor	Value	Press	No	Angle
(-)	(-)	(-)	(-)	[m]	[m]	[-]	[-]	[Pa]	[-]	[deg]
L1	*CR1*	*-1*	*kit*	*1.1*	*1.1*					
L2	*CR2*	*kit*	*hoo*	*1.5*	*1.5*					
L3	*FA1*	*hoo*	*-2*	*3*	*3*	*d*	*800*	*d*	*FA1*	
L4	*RF1*	*hoo*	*kit*	*2.5*	*2.5*	*d*	*d*	*d*	*L3*	

The fan has a normal speed of 800 rpm and a schedule FA1, which switches the fan in different positions. To be able to use link L3 as reference, this reference must appear before the RF that is using it, or else the reference flow rate would not be known or contain an old value at the time the RF flowrate is calculated.

It would be better to use these RF types always in pairs to have balanced flow, like in the next example.

&-NET-LINks 22

Link	Type	Zone ID		Height		Height	Factor/ Actual RPM/	3Dflow or	Schedule Name (5 Char.)	
									T- Junct.	Ref. Link
No	Name	From	To	From	To	Factor	Value	Press	No	Angle
(-)	(-)	(-)	(-)	[m]	[m]	[-]	[-]	[Pa]	[-]	[deg]
L1	*CR1*	*-1*	*kit*	*1.1*	*1.1*					
L2	*CR2*	*kit*	*hoo*	*1.5*	*1.5*					
L3	*FA1*	*hoo*	*-2*	*3*	*3*	*d*	*800*	*d*	*FA1*	
L4	*RF1*	*hoo*	*kit*	*2.5*	*2.5*	*d*	*d*	*d*	*L3*	
L5	*RF1*	*kit*	*hoo*	*2.5*	*2.5*	*d*	*d*	*d*	*L3*	

To avoid convergence problems, RF has no flow rate derivative. This means that a zone cannot be linked only with RF types, there must also exist some normal links, like CR or FA.

This results in flows in both directions and will have no effect on the flow balance during the iteration process. It will result in the desired flow exchange that can spread pollutants.

5.4.4.2 Description of a Passive Stack in the Network

&-NET-LINks 22

Link	Type	Zone ID		Height		Height	Factor/ Actual RPM/	3Dflow or	Schedule Name (5 Char.)	
									T-Junct.	Ref. Link
No	Name	From	To	From	To	Factor	Value	Press	No	Angle
(-)	(-)	(-)	(-)	[m]	[m]	[-]	[-]	[Pa]	[-]	[deg]

Example Input:

33 PSround KIT -cpS 2.4 4
34 PSrect WC1 -cpS 2.4 4

Here cpS is the external pressure point at the position of the outlet of the duct above the roof.

Schedules help to allow changing certain parameters in time (e.g., the weather). If the schedule has no event at the start-time of the simulation, COMIS will use the value at the last event before the start-time. If the schedule starts with an event after the start-time, the schedule values are set to the defaults provided by the program.

5.5.1 [Temporarely Removed]

5.5.2 [Temporarely Removed]

5.5.3 Window Schedule

Keyword:

&-SCH-WINdow schedules

Header:

Schedule *Name (-)	Time (-)	Opening Fraction (-)

Example input:

**W1*	*06:00*	*.50*
	06:10	*1.000*
	06:15	*.0*
	08:00	*.10*
	12:00	*.30*
	18:40	*.0*

or

**W2*	*FRI_06:00*	*1.00*
	FRI_07:00	*.0*
	07:00	*.40*
	08:00	*.0*

or if the data are defined in a separate file:

F: W3 window1.dat

Description:		
Parameters	Description	Input Format
Sched. *Name	Name of schedule name must be preceded by '*' and begin with 'W'. For data file assignment the following format must be used: F: <schedule name> <file name.ext>	string <= 10 char
Time	Date_time string	
Opening Frac..	Opening fraction of window (see window air flow component definition)	Real 0.0..1.0

In the window schedule the opening fraction of a window can be changed. The schedule name must be specified in the last column of the section &-NET-LINKS in the lines with the corresponding windows. The value under “Factor/Actual RPM/Actual Value” of this line will then be changed according to the schedule.

5.5.4 Fan Schedule

Keyword:

&-SCH-FAN schedules

Header:

Schedule *Name (-)	Time (-)	Fan Speed Factor (-)

Example input:

```
*F1     FRI     1.1
        SAT     0.8
        SUN     0.75
        MON     1.4

*F2     06:00   1.0
        07:00   0.7
        18:00   1.18

*F3     WKD     1.0
        WDY     0.5
```

or if the data are defined in a separate file:

F: F3   Fan3.Dat

Description:

Parameters	Description	Input Format
Sched. *Name	Name of schedule; name must be preceded by '*' and begin with 'F'. For data file assignment the following format must be used: F: <schedule name> <file name.ext>	string <= 10 char
Time	Date_time string	
Fan Speed Factor	Factor of actual to nominal fan speed	Real >=0.0

Similar to the window schedule, the fan schedule is connected to the air flow component it belongs to by specifying the schedule name in the &-NET-LINKS description. The fan speed factor of the schedule will replace the "Actual RPM" in **&-NET-LINks.**

5.5.5 Temperature Schedule

Keyword:

&-SCH-TEMperature schedules

Header:

Schedule *Name	Time	Temp
(-)	(-)	(°C)

Example input:

**T1*	*SAT*	*17*
	MON	*20*

F: T2 Temp.Dat

Description:

Parameters	Description	Input Format
Sched. *Name	Name of schedule name must be preceded by '*' and begin with 'T'. For data file assignment the following format must be used: F: <schedule name> <file name.ext>	string <= 10 char
Time	Date_time string	
Temp	Room temperature value	Real

Every time the program reads a line of a temperature schedule it searches in the last column of the &-NET-ZONes description for the zones with the respective schedule name. In these zones the temperature (Tz) is then changed.

5.5.6 Humidity Schedule

Keyword:

&-SCH-HUMidity schedules

Header:

Schedule *Name (-)	Time (-)	Humidity (g/kg)

Example input:

**H1*	*07:00*	*12*
	07:40	*10*
	08:00	*11*

or:

F: H2 Hum2.Dat

Description:		
Parameters	Description	Input Format
Sched. *Name	Name of schedule; name must be preceded by '*' and begin with 'H'. For data file assignment the following format must be used: F: <schedule name> <file name.ext>	string <= 10 char
Time	Date_time string	
Humidity	Humidity in a zone, replacing the value given in the zone definition data section	Real >=0.0

When the program reads a line of a humidity schedule it searches in the last column of the &-NET-ZONes description for the zones with the respective schedule name. In these zones the humidity (Xhz) is changed.

5.5.7 Sink Schedule

Keyword:

&-SCH-SINk schedules

Header:

Schedule *Name (-)	Time (-)	Sink Factor (-)

Example input:

**S-H2O 12:00 0.5*

or

F: S-CO2 Sink.Dat

Description:

Parameters	Description	Input Format
Sched. *Name	Name of schedule; name must be preceded by '*' and begin with 'S'. For data file assignment the following format must be used: F: <schedule name> <file name.ext>	string <= 10 char
Time	Date_time string	
Sink Factor	Factor of actual value to value given in Zone pollutants data block for pollutant No 1 (5.4.3.3)	Real >=0.0

Data lines in the sink schedule are processed in the following way: The program checks for the link schedule name in the last column of the keyword **&-NET-ZONes**. For each zone for which the program finds this schedule name it changes in the pollutant description (keyword &-NET-ZP) the value of the sink strength of the first pollutant by multiplying this value with the sink factor given in the sink schedule.

For more details, please see 5.5.8 Source Schedule.

5.5.8 Source Schedule

Keyword:

&-SCH-SOUrce schedules

Header:

Schedule *Name (-)	Time (-)	Source Factor or Number of Occupants (-)

Example input:

**Qh2o*	*00:40*	*1.0*
	01:40	*0.1*
**OCC-CO2*	*00:00*	*2*
	09:00	*40*
	12:00	*2*

or

F: Qco2 OccSou.Dat

Description:

Parameters	Description	Input Format
Sched. *Name	Name of schedule; name must be preceded by '*' and begin with 'Q'. for source data or with 'OCC' if source are occupants For data file assignment the following format must be used: F: <schedule name> <file name.ext>	string <= 10 char
Time	Date_time string	
Source Factor	If Source schedule is specified as 'Q' type: Factor of actual value to value defined for Pollutant No 1 in zone pollutant data block (5.4.3.3)	Real >=0.0
Number of Occupants	If Source schedule 'OCC' is specified as 'OCC' type: Number of occupants specified in the first line in the occupant description data block	Integ >=0

If the name of the source schedule starts with a 'Q', which is an indicator for source data, the program searches for the source schedule name in the last column of the keyword **&-**

NET-ZONes. For each zone for which it finds this schedule name it multiplies the source strength value of the first pollutant in the pollutant description (keyword &-NET-ZP) with the source factor of the schedule. This is the same procedure as processing the sink schedule.

If the name of the source schedule starts with 'OCC', the last number of the schedule line is the number of occupants in the zones to which the schedule belongs.

5.5.8.1 The use of Sources and schedules

Sources appear under the following keywords:
&-PR-UNITs
&-NET-ZONes
&-NET-ZP zone-pollutants
&-SCH-SOUrce schedules
&-POL-DEScription

It is possible to have:
more than one source schedule per zone.
more schedules using the same pollutant
longer names at &-POL-DEScription than in the schedules

To make use of the same schedule for a pollutant in more than one zone but with source strengths that differ per zone (for instance proportional to the room area) it is sufficient to give different nominal source strengths at NET- ZP zone-pollutants.

In section **&-PR-UNITs** one unit of all Nconc (normal maximum=5) sources can be given. Under keyword **&-NET-ZONes** sources may appear in the last column, Schedule name. The source schedule has to start with Q, followed by an optional number, followed by the first part of the pollutant name. A pollutant name must start with a character A..Z,a..z.

The format for the source schedule name is:
Q[number]polname
Q = mandatory letter
number = may be used to make more schedules for one and the same pollutant substance
polname= first part of a pollutant name

A few examples are:
Qpol1
QCO
Q23pol1

Under the section **&-NET-ZP** zone-pollutants the initial (start) values for pollutant concentrations can be given with the nominal value of source strength and sink strength. The first line (with the zone name identifier) is for the first pollutant (as defined at &-POL-DEScription). The second line for the second pollutant, etc.

&-SCH-SOUrce schedules
The first line of a schedule has the Source schedule name as identifier preceded by * . The schedule name includes the starting Q and should match exactly the name given at &-NET-ZONes. The identifier is followed by a time and a multiplication factor for the nominal source strength given at &-NET-ZP zone-pollutants

&-POL-DEScription has 3 columns:
1- the pollutant sequence number (normally 1..5),
2- the pollutant name which may be longer than the fraction used at the
 corresponding Source schedule name,
3- the molar mass

The sequence number corresponds with the sequence at &-NET-ZP zone-pollutants. To match the schedule Qpol1 a pollutant name must start with pol1, therefore: pol1, pol1gas etc are OK . The name can be longer to be more descriptive while the schedule name can be short.

Occupants also may have a source contribution. Until now CO2 and O2 are interpreted as occupant sources. H2O and C3H6O will follow. These source only come into play if the pollutants defined in the simulation (POL-DES) match (the same way as source schedules). So a pollutant must be CO2-etc-etc and then the occupant source CO2 (if defined in the continuation lines of '**OCCUPANt** definition') will actually be added to the zones this occupant is in. An occupant is in a zone if the activity in that zone of that occupant is >0 .

The source of an occupant in a zone is OccPol(Occnr,1..2)*OccAct(OccNr,ZoneNr) factor 1..1.8..3

The activity of an occupant in a zone is OccAct(Occnr)*OccAct(OccNr,ZoneNr)

At the occupant schedules 'activity factor' and 'number of occupants' is multplied and stored in OccAct(*). For older schedules one may directly put the product in the 'activity factor' and not use the 'number of occupants' To remove a person from a room one has to set the 'activity factor' and 'number' to their new values.

The Occupant schedules move occupants to a zone by their sequence number! Changing the zone sequence will change the location where the occupants will reside.

5.5.9 Occupant Schedule

Keyword:

&-SCH-OCCupant schedules

Header:

Schedule *Name (-)	Time (-)	Zone ID (-)	Activity Level Number Factor (-)

Example input:

**OCC1*	*07:00*	*1*	*0.60*
	07:30	*2/3/5*	*0.10*
	07:50	*9/1to4/8*	*0.00*
	08:00	*ALL*	*0.90*
	10:00	*ALL7/8*	*0.80*
	11:00	*ALL1*	

or

F: OCC2 Occup.dat

Caution: The Zone ID “# ALL” has not been implemented yet

Description:

Parameters	Description	Input Format
Sched. *Name	Name of schedule; name must be preceded by '*' and begin with 'OCC' . For data file assignment the following format must be used: F: <schedule name> <file name.ext>	string <= 10 char
Time	Date_time string	
Zone ID	Zone Identification in which the occupant is located. Zone must be defined.	
Number of Occupants	If Source schedule 'OCC' is specified as 'OCC' type: Number of occupants specified in the first line in the occupant description data block	
Activity Level Factor	Factor actual activity to the one specified in the occupant description data section.	Real >=0.0

All data given refer to the first occupant defined in the occupant description data section.

With the occupant schedule the activity level of an occupant type in a specific zone can be changed. In the occupant description up to 5 occupant types can be specified. Each occupant type has an activity level, which is the general activity of this occupant type. With the help of the occupant schedule one now can change the activity of a selected occupant type in one specific zone, so that one can have occupants of the same type but with different activity levels in different zones of the building.

To which occupant type the schedule entry refers is determined by the name of the schedule. The schedule name must be OCC1 to OCC5 (preceded by '*' for indicating a name) which means that the schedule data belongs to occupant type number 1 to 5. The original activity level of this occupant type is then multiplied with the factor given in the schedule line, but only for the occupants in the zone specified in the schedule.

As long as one does not use an occupant schedule, an occupant type as defined in &-OCCUPANt description is not attached to a zone. Therefore one needs an occupant schedule if one has an occupant description where is specified in which zones this occupant can be found. By using occupant schedules it can be distinguished between different activity levels of one occupant type in different zones. Since it is not very convenient to write a new line for each zone in which is an occupant of a particular type, it is possible to specify the affected zones in a list (like in the main schedule).

The default values are 'ALL' for the list of zones and 1.0 for the activity level factor. An activity level factor of 1.0 means that the original activity is used again.

Caution: The Zone ID “# ALL” has not been implemented yet

5.5.10 Multi-Schedule

Keyword:

&-SCH-MULTI schedules

Header:

1.	Multi-Schedule Filename

Example input:

F: multi.dat

Description:		
Parameters	Description	Input Format
Filename	Name of the Multi Schedule File F: <file name.ext>	string

2.	Schedule 1 Name	Schedule 2 Name	10 Name

Example input:

t.house *INVALID* *t.attic*

Description:		
Parameters	Description	Input Format
Name	Name of the schedule in the appropriate column. The type of the schedule is determined by the name. If the name is 'INVALID' the column is skipped. The names are case sensitive.	string <= 10 char. All schedule names including the separators (blanks) should not exceed 120 characters Max. number of schedules = 10

The Multi-Schedule is used to avoid having a separate file for each schedule. Also, since the data in the Multi-Schedule file has to be sorted, no sorting needs to be done during the program run.

Schedules supported by Multi-Schedule are:

Window (W), Fan (FA), Temperature (T), Humidity (H), Sink (S), Source (Q), Meteo (METEO) and Pollutant.

The schedule names have to follow the conventions defined in the preceding chapters. However, the Meteo Schedule spawns over several columns in the Multi-Schedule file:

Wind speed Wind Direction Temperature Humidity Pressure

The Pollutant Schedule uses the names 'pol1' to 'pol5' for the outdoor pollutants 1 to 5.

The Multi-Schedule file has a form similar to the COMIS weather file (see Section 5.7.3). In the first line is an asterisk, optionally followed by the start time and date and the interval length. If start time and date are given, the asterisk has to be followed by a blank. The next line gives the names of the schedules. These have to match the names in the CIF ('INVALID' in the CIF matches any name, however). After this header the schedule data follows.

Example input:

```
*  1992jan01_0:00 3600
t.house        pol1        t.attic
20.1           .1          18.3
23.4           .3          19.1
```

NOTE: Only one multi-schedule file is allowed.

5.6 *Cp-Values*

5.6.1 Building Reference Height for Cp Data

Keyword:

&-CP-BUILding reference height for Cp data

Header:

Height (m)

Example input:
10.0

Description:			
Parameters	Description	Input Format	Default
Height	Building reference height for related cp-value data	Real >0.0	10.0

5.6.2 Cp-Value Input

Keyword:

&-CP-VALUes

Header:

1.	Dataset Name

Example input:

SIMPLE_HOUSE_1

or if data file assignment:

F: leso.cp1

Description:		
Parameters	Description	Input Format
Dataset name	Any name For data file assignment: F:<filename>	String <80 char

Header:

2.	Facade	Wind Direction	(first line)						
	Elem.no	Cp Values (second and following lines)							
	* (-)	(deg)	[deg]	[deg]	[deg]	[deg]	[deg]	[deg]	[deg]

Example input:

*	*0*	*90*	*180*	*270*
4	*1.0*	*-.49*	*-.45*	*-.49*
1	*-.49*	*-.45*	*-.49*	*1.0*
3	*-.45*	*-.49*	*1.0*	*-.49*
2	*-.49*	*1.0*	*-.49*	*-.45*

More than one block may be given, thus allowing for more wind-direction columns. The facade element numbers must be the same in each block Each block can contain eight different wind directions.

2.	Facade Elem.no	Wind Direction (first line) Cp Values (second and following lines)							
	* (-)	(deg)	[deg]	[deg]	[deg]	[deg]	[deg]	[deg]	[deg]

Example input:

*	*351*	*352*	*353*
4	*-.49*	*-.45*	*-.49*
1	*-.45*	*-.49*	*1.0*
3	*-.49*	*1.0*	*-.49*
2	*1.0*	*-.49*	*-.45*

Description

Parameters	Description	Input Format	Default
Facade Elemno	Number of facade element related to the given Cp value data line	Int >0 -	
Wind Direction	The first data line contains the wind-direction values. The following Cp values are related to these directions.	Real 0.0 ...360.0	0.0
Cp Value	Cp value	Real	0.0

5.6.3. Cp-Calculation Routines

5.6.3.1 General

5.6.3.2 Description of the Routines

These routines calculate Cp at any position of a surface element on the envelope of a block-shaped building for given values of various environmental and geometric parameters.

According to application limits related to the variation range defined for each parameter the routines cannot deal with :

- high terrain roughness (VeEXP > 0.33)
- surroundings with staggered or irregular pattern layout
- downtown (PAD > 0.5)
- urban environment (PAD > 0.125) when the building to be considered has a different height from its surroundings
- buildings 4 times higher or 0.5 lower than the surroundings
- buildings with irregular shape or overhangs
- regular block-shaped buildings with aspect ratios rather far from a cube (0.5 > LX/LZ > 4.0 , 0.5 > LY/LZ > 2.0)

5.6.3.3 Input Parameter

5.6.3.3.1 Building Rough Outside Dimension

Keyword:

&-CPR-BUIlding data

Header:

Building Rough Outside Size (block model) for Facade Element Positioning		
LX (m)	LY (m)	LZ (m)

Example input:

25 *30* *12*

Description		
Parameters	Description	Input Format
LX, LY, LZ	Dimensions of the building block along x, y and z axes	Real > 0.0

5.6.3.3.2 Facade Element Position Data

Facades have fixed orientations on the block: 1,2,3,4 are walls 1=front 3=back 2=right 4=left (facing the front wall) 5=roof 6=floor.

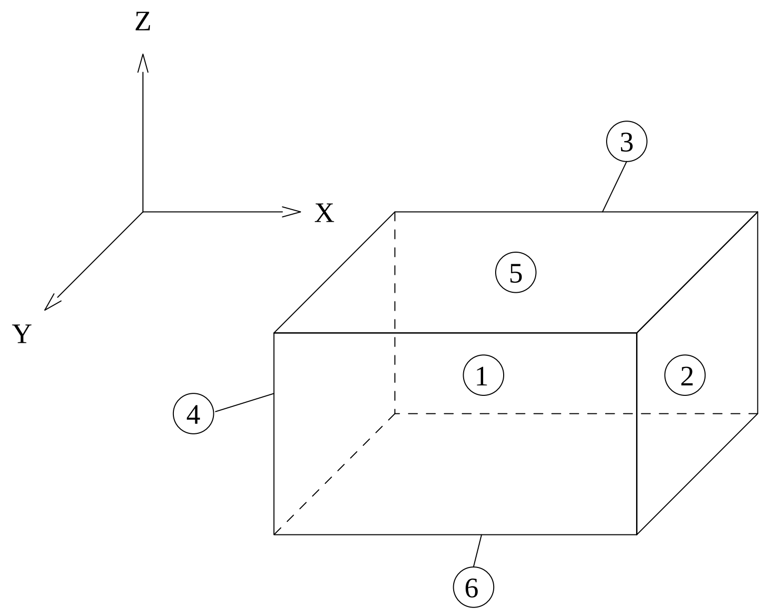

Figure 5.6.1: *Facade Numbering*

On each facade, a surface element is defined by its horizontal and vertical relative position on the facade seen as lying on a vertical plane with coordinate reference at the lower-left corner as facing the building.

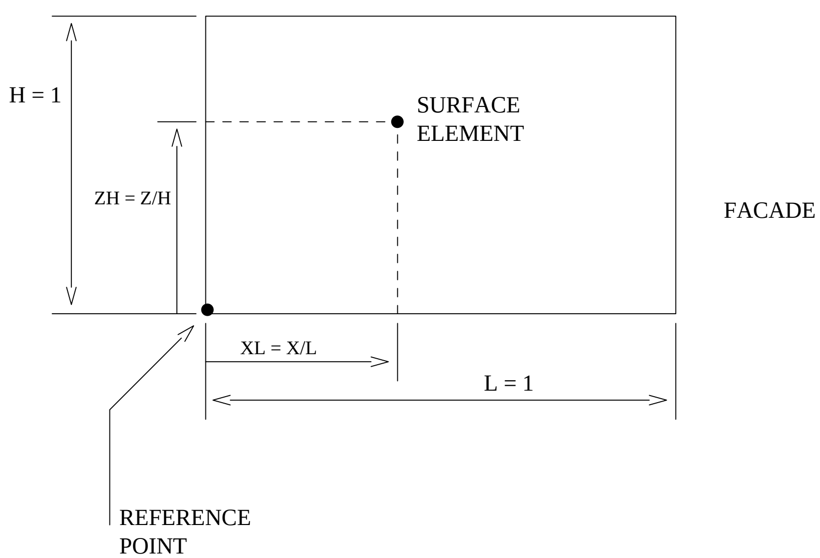

Figure 5.6.2: *Facade Element Positioning*

Keyword:

&-CPR-FACade elements definition

Header

Facade *No	Facade Element No	XL	ZH
(-)	(-)	(-)	(-)

Example input

**1*	*10*	*0.3*	*0.5*
	11	*0.3*	*0.7*
	12	*0.7*	*0.5*
	13	*0.7*	*0.7*
**2*	*21*	*0.5*	*0.2*
	22	*0.5*	*0.7*
**3*	*31*	*0.3*	*0.5*
	32	*0.3*	*0.7*
	33	*0.7*	*0.5*
	34	*0.7*	*0.7*
**4*	*41*	*0.5*	*0.2*
	42	*0.5*	*0.7*

Description

Parameters	Description	Input Format
Facade *No	Number of the facade on which the external surface element is located(see Fig. 5.6.1)	*<No> No: INT between 1...6
Facade Element No	Number of the individual Facade element. This number corresponds to the number used in the network-zones and in the cp value input	INT > 0
XL	Relative horizontal position (x/l) of the surface element measured, facing the facade, from the left edge (see Fig. 5.6.2)	0.0 _< REAL _< 1.0
ZH	relative vertical position (z/h) of the surface element measured (in case of facade 5 and 6 the bottom edge coincides with the upper and the bottom edge, respectively, of the facade 1) (see Fig. 5.6.2)	0.0 _< REAL _< 1.0[4]

[4] in case of an isolated from the bottom edge of the facade cube, at VeEXP=0.22: 0.1 _< REAL _< 0.9; if VeEXP =/= .22): 0.07 _< REAL _< 0.93(all other cases0)

5.6.3.3.3 Wind-Direction Data

Keyword:

&-CPR-WINd-directions

Header:

Wind directions									
(deg)	[deg]	[deg]	[deg]	[deg]	[deg]	[deg]	[deg]	[deg]	[deg]

Example input:

0 90 180 270

Description		
Parameters	Description	Input Format
Wind-Directions	Wind direction values for which cp-data are to be calculated. At least one value has to be given	0.0 _< REAL < 360.0
	Wind angles are measured clockwise from North semi-axis and to each of them a value for the environmental parameters as described in Section 5.7.2 has to be given	

5.7. Environment Description

5.7.1 Building Related Parameters

Keyword:

&-ENV-BUIlding related parameters

Header:

1. Altitude	Angle Building	Geographic Position	
	North to -X-Axis	Latitude + = N	Longitude + = E
(m)	(deg)	[deg] - = S	[deg] - = W

Example input:

588.0	*34.04*	*7.65*	*10.0*

Description			
Parameters	Description	Input Format	Default
Altitude	Altitude of building entrance above sea level	Real	0.0
Angle Building	Angle from North-semiaxis to the building, X-semiaxis, measured clockwise (see Fig. 5.7.1).	Real 0.0...360.0	0.0
Latitude	Latitude >0 : Northern hemisphere Latitude <0 : Southern hemisphere	Real 0.0... 90.0 Real 0.0... 90.0	43.0
Longitude	Longitude >0 : East Longitude <0 : West	Real 0.0...180.0 Real 0.0..-180.0	0.0

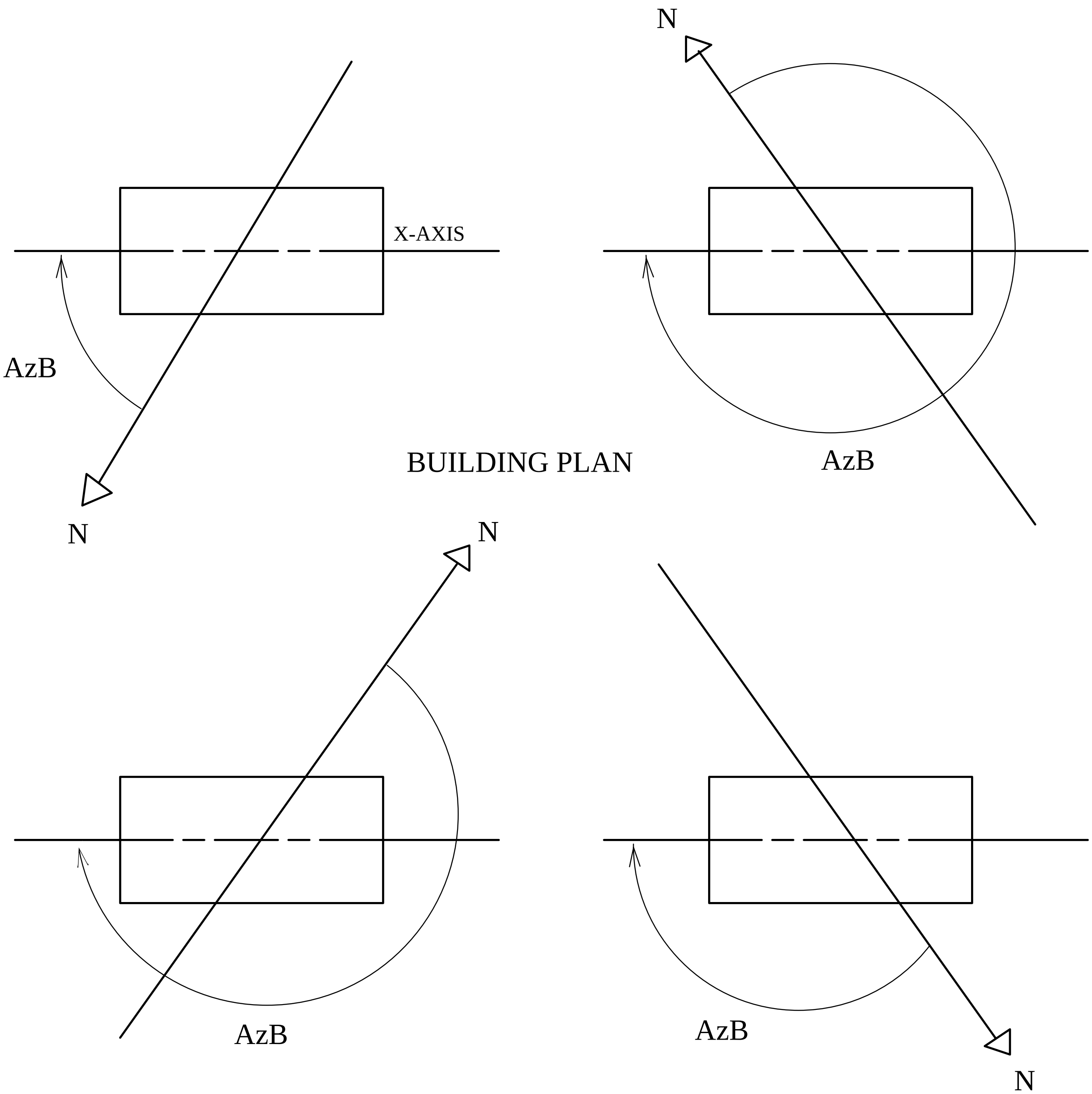

Figure 5.7.1: *Azimuth Angle of a Building*

5.7.2 Wind and Meteo Data related Parameters

COMIS supports the two main approaches for wind velocity profiles, the Power-Law profile and the Logarithmic profile. At the &-UNITS keyword in the *.CIF file the choice can be made for the profile to be used.

Example:

&-UNITS

```
#  Name      input    output
   profile  z0       alpha
```

No matter which profile is taken, the validity for urban areas is usually small, unless the building is taller than its surrounding. In case of low buildings in rough terrain and large surrounding obstacles, direct Cp data from wind tunnels or on site measurements can be used.

The program uses the given wind speed at the meteo site. It calculates the speed at 60 m high (or higher if meteo or the building is in rough terrain). This speed at 60m (or higher) is assumed to be equal to the wind speed at the same height above the building. Along the profile near the building the velocity at the CP-BUILding reference height is calculated.

From this it is clear that the wind velocity at CP-building-reference height is not a direct input.

Keyword:

&-ENV-WINd and meteo data related parameters

Header:

1.	Ref. Height for Wind Speed (m)	Altitude Meteo Station (m)	Wind Velocity Profile Exponent (-)

Example input:

	10.0	*456.00*	*0.14*

Description			
Parameters	Description	Input Format	Default
Ref. Height for Wind Speed	Reference height for wind speed meteo data	Real >0.0	10.0
Altitude Meteo Station	Altitude of meteo station	Real	0.0
Wind Velocity Profile Exponent	Exponent in the power law function of the wind mean velocity profile.	Real	0.14

Header:

2.	Wind Direction Angle (deg)	Plan Area Density (-)	Wind Velocity Profile Exponent (-)	Surrounding Building Height (m)

Example Input :

0	*.45*	*.22*	*12.5*
45	*.23*	*.16*	*7.0*
90	*.32*	*.33*	*16.0*
175	*.05*	*.28*	*4.5*

Description

Parameters	Description	Input Format	Default
Wind Direction, Wind Angle	Wind angle direction for related data measured clockwise from geographic North	0.<=Real<=360.	0.0
Plan Area Density†	Factor for defining the obstruction density of surrounding buildings (PAD)(see Fig. 5.7.2) (block models with same height, normal layout pattern); the reference area surrounding the building spans from a radius of 10 to 25 times the building's height, inversely to the density	0.<=Real<=0.5	0.0
Wind Velocity Profile Exponent	Exponent in the power law function of the wind mean velocity profile Reference value: 0.10 level surface with very small obstructions, grassland 0.22 rolling or level surface broken by numerous obstructions, such as trees or small houses 0.32 heterogeneous surface with structures larger than one story	Real	0.18
Surrounding Buildings Height†	Surrounding buildings rough eave height, (SbH)(block models with same height, normal layout pattern); in case of PAD = 0, SbH has to be taken as the height of the building	Real > 0.0	0.0

†Only being used for Cp calculation routines; use "0.0" for all other cases

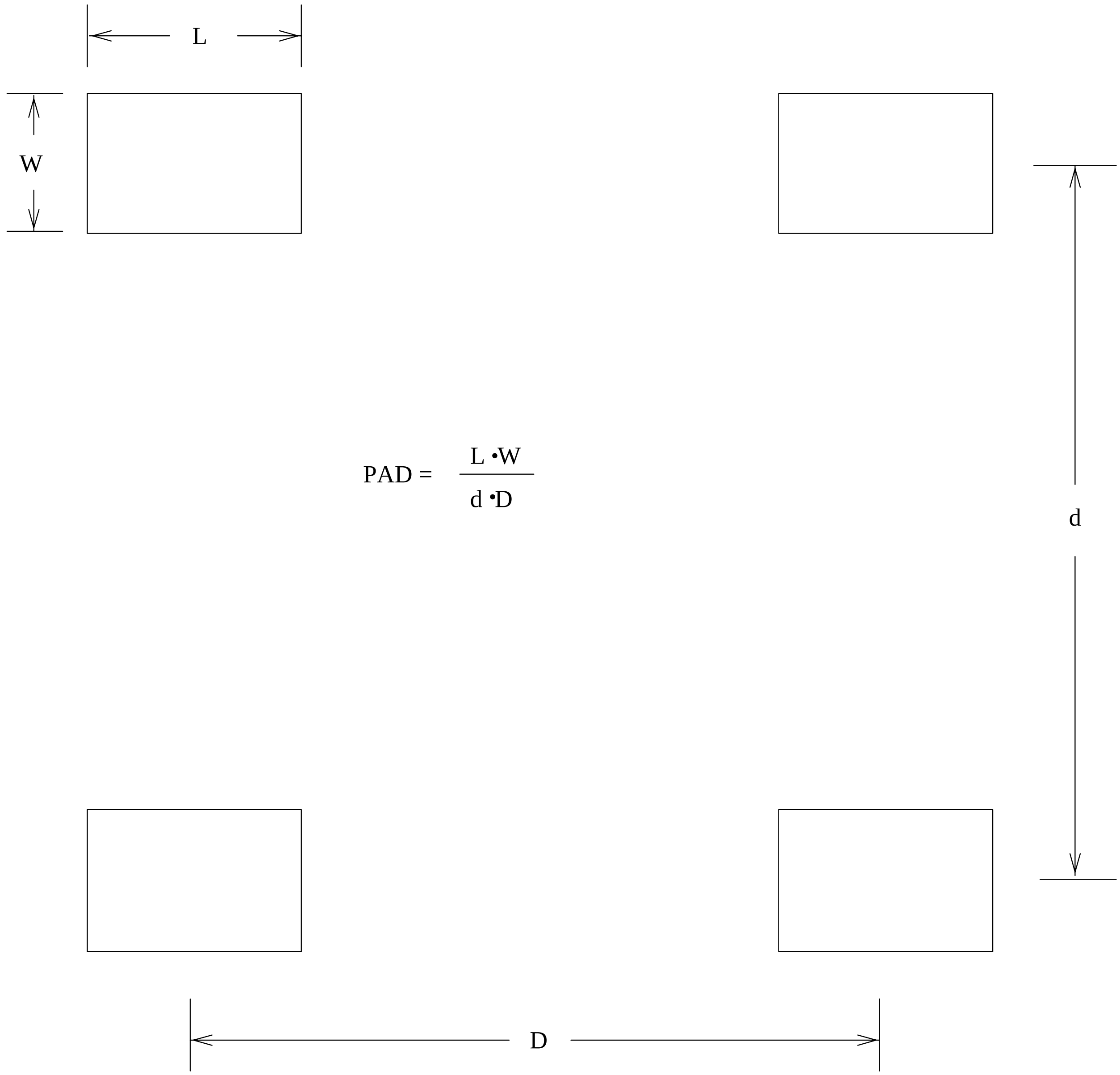

Figure 5.7.2: *Plan Area Density (only necessary for Cp calculation)*

5.7.3 Meteo Data

Keyword:

&-SCH-METeo data

Header:

1.	Dataset Name

Example input:

METEO_typical_for_here

or, if file assigned:

F: berk87.met COMIS

Description		
Parameters	Description	Input Format
Dataset name	Any name For data file assignment: F:<filename> <format	String <80 char format= COMIS or DOE2

Header:

2.	Time	Wind		Temperature	Humidity	Barometric Pressure
		Speed	Direction			Absolute
	(-)	(m/sec)	(deg)	(°C)	[g/kg]	[kPa]†

Example input:

1968may23_08:30:10 2.0 10 15.1 8.4 98.80

Description			
Parameters	**Description**	**Input Format**	**Default**
Time	Date_Time string	(see § 5.1.2)	jan01_
Wind Speed	Wind speed at ref. height	Real>0.0	0.0
Wind Direction	Wind direction at ref. height	Real 0.0..360	0.0
Temperature	Dry bulb temperature	Real	20.0
Humidity	Absolute humidity	Real	0.0
Pressure	Absolute barometric pressure at meteo station†	Real>0	101.325

†The barometric pressure must be in kPa.

The meteo data can be stored under the keyword &-SCH-MET in the CIF input file or in a separate weather file. When using a weather file, the data under the first header of the meteo schedule must contain a data file assignment, marked by an 'F:' followed by the respective filename. When both, separate weather file and meteo description in the CIF file, are used and there are entries for the same time in both files, the CIF meteo description 'overwrites' the weather file data.

Several formats are allowed for the weather file.

1. COMIS format (default)

The file can contain data of the same format as the data which would be written directly into the CIF input file, that is, every data line starts with a time string followed by the information about wind speed, wind direction, etc. If there are data missing, the program assumes default values.

If one has a lot of weather data which change regularly, a more convenient way of inputting the data is, not to specify the current time at the beginning of each line, but to begin the file with a line which specifies the start time and the time interval between the data lines. The first character of this line must be an asterisk ('*') to mark that this kind of format is used. It is not necessary to specify both parameters, when one wants to accept the defaults: start time = Jan 01 and hourly data changes.

The weather file may contain blank lines, which are ignored by the program.

Since the weather file is sequentially read by the program, the time statement must at least consist of month and day specification. If the year is omitted, the year of the start of the simulation interval is assumed. A time statement only containing a time and no date cannot be processed in that way that the program reads the data of this line every day again.

Example input for the weather file in COMIS format:

1991may23_08:30:10	*2.0*	*10*	*15.1*	*8.4*	*98.80*
1991may23_09:35:30	*1.8*	*11*	*15.1*	*8.4*	*97.00*

or

**1991may23_10:00   3600*
2.0   10   15.1   8.4   98.80
2.3   11   16.0   8.4   97.00

or

**1991jun01_*
2.0   10   15.1   8.4   98.80
2.3   11   16.0   8.4   97.00

or

*
2.0   10   15.1   8.4   98.80
2.3   111   6.0   8.4   97.00

2. DOE2 format

It is also possible to use weather files in the (packed) DOE-2 format. This has the advantage that not two different weather files need to be stored if the user runs both, DOE-2 and COMIS simulations.

The program expects the weather data to be in the DOE-2 format when the keyword "DOE2" is specified after the filename.

Example input for header 1 to assign a file in DOE-2 format:

F: zagreb.bin DOE2

5.7.4 Pollutants

5.7.4.1 Pollutant Description

Keyword:

&-POL-DEScription

Header:

No	Name	Molar Mass
(-)	(-)	(g)

Example input:

1	*CO2*	*44*
2	*FORMALDEHYD*	*30*

Description			
Parameters	Description	Input Format	Default
No	Number of pollutant	Integer 1...5	jan01_
Name	Name of pollutant	String <20 char	pollutantx
Molar Mass	Molar mass of respective pollutant	Real >0.0	28.6

5.7.4.2 Pollutant Description

Keyword:

&-HISTO

The types of histograms are defined in a new keyword section. If Concentrations are used in the histogram, then this section must come after &-POL-DEScription ! For details see §5.2.2.

5.7.4.3 Outdoor Pollutant Concentration Schedule

Keyword:

&-SCH-POL outdoor concentration data

Header:

1.	Dataset Name

Example input:

Typical_Pollutant_1

or in case of a pollutant file assignment:

F: examp88.poll

Description		
Parameters	Description	Input Format
Dataset name	Any name For data file assignment: F:<filename>	String <80 char

Header:

2. Time	Pollutant Concentration				
	No1	No2	No3	No4	No5
(-)	(kg/kg)	[ ]	[ ]	[ ]	[ ]

Example input:

0101_00:00 0. 0. 0. 0. 0.
10:23:59 fef 1=.011 53=0.0 23to35=.12
10:24:59 fef .011 22(0.0) 13(0.12)

In this example 13(0.12) will be interpreted as 13 values of 0.12

Description		
Parameters	**Description**	**Input Format**
Time	Date_time string	(see § 5.1.2)
Pollutant Concentration	Pollutant outdoor concentration In the schedule of outside concentrations one may also define another factor that is multiplied with the outdoor concentration to get the concentration at the facade element (=fef). In this case the input format is as follows: FEF <FeNr>=<concentration value> or: FEF <concentr. value> <concentr. value> .....<concentr. value> for all facade elements	Real >=0.0

(1) If "fef" is not specified, the given concentrations for the pollutants 1 to 5 refer to all external nodes. That means, for every external node (=> keyword &-NET-EXTERNAL-NODE-DATA) the schedule concentration of pollutant no. 1 is multiplied with the outdoor concentration factor OuCF(i) (third column of **&-NET-EXT**) of external node i. The same is done with the pollutants 2 to 5. If a concentration is not found for all of the pollutants in the pollutant schedule, the program takes 0.0 as a default.

(2) If the program finds the "fef"-keyword, the given values are taken as a factor to the current outdoor concentration at this facade. Therefore, the program checks for all external nodes connected with the respective facade element and multiplies the correspondent current values of the outside concentrations for all 5 pollutants with the factor found in the schedule.

5.8 Occupant Description

Keyword:

&-OCCUPANt description 39 -OPTIONAL DATA SECTION-

Header:

No	Sex	Age	Height	Mass	Activity	Cigarets
(-)	(-)	(a)	(m)	(kg)	(W/m2)	[1/h]

Example input:

1 MALE 5 1.03 15 60 0

Description			
Parameters	Description	Input Format	Default
No	Number of occupant	Integer 1...5	-
Sex	Keyword for sex:	Keyword string	MIX
	MALE MIX		
	FEM:ale		
Age	Age	Real >0.0	20
Height	Height	Real >0.0	1.70
Mass	Body mass of occupant	Real >0.0	70.0
Activity	Activity (level)	Real >0.0	58.0
Cigarets	Average number of cigarets the specific occupant smokes per h	Real >=0.0	0.0

6. Input Example

Sections 6 and 7 are describing the input and output of the COMIS model. To make things somewhat clearer, this section uses an example based on a real building to demonstrate the input and the output procedures.

6.1 Building Description

The example is based on measurements on a building at the Richmond Field Station at University of California at Berkeley, California, USA. The measurements were part of the COMIS project. The measurements were performed in Fall 1989 and the evaluation was made at INSA de Lyon in France Fall 1990.

Figure 6.1 A-B shows the layout of the building. It is a two story, 6-zone wooden structure. The overall Specific Leakage Area is approximately 6.5 $cm^2/m^2_{floor\ area}$. The building's floor plan includes the orientation of the building, the numbering of the facades and all flow paths. The roof (r) is given the facade number 5. Table 6.1 shows the leakage coefficients and exponents determined at INSA de Lyon.

DRxx stands for internal doors while RMxx stands for outer surfaces facing different directions, N, E, S, and W. The individual windows and outer doors, as shown in Figure 6.1 A-B, are not used. Just one leakage function was determined for the external walls for each room. The leakage coefficient has been distributed according to the areas of different walls facing different directions. Furthermore, measured data for DR34 is missing as well as all vertical leakages except the staircase. These values have been estimated based on the other internal leakages.

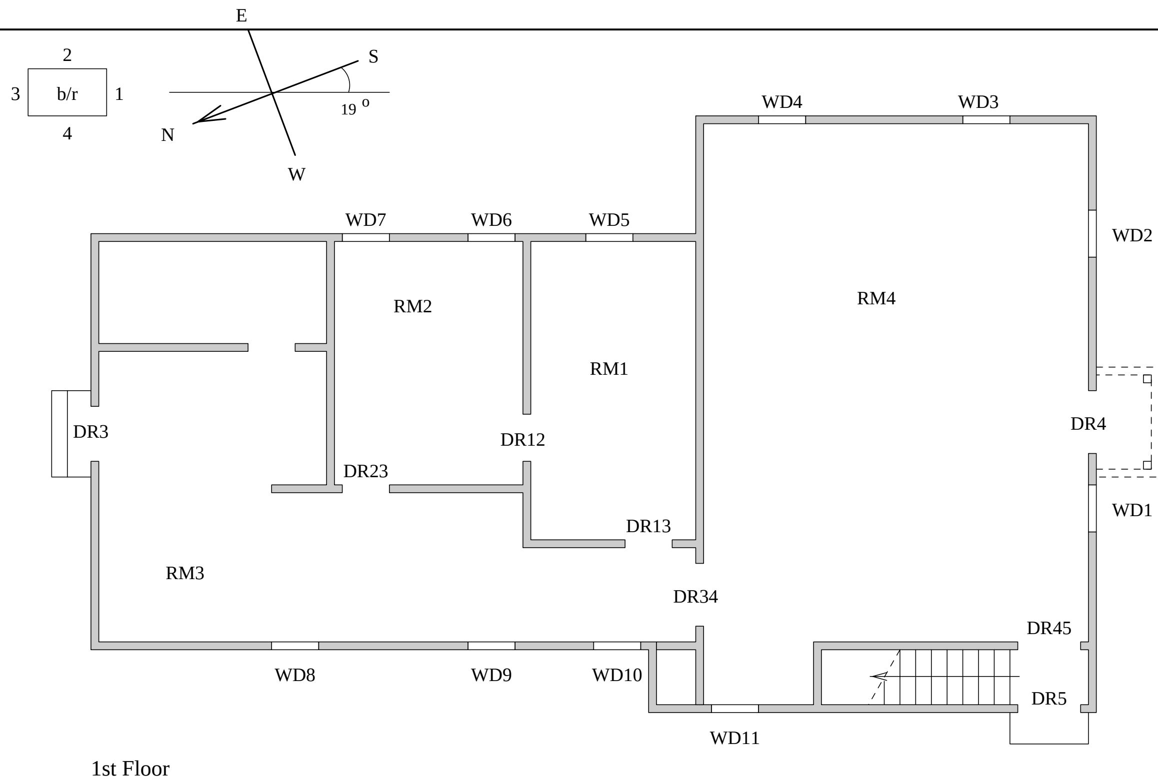

Figure 6.1 A: *Layout of the 1st Floor*

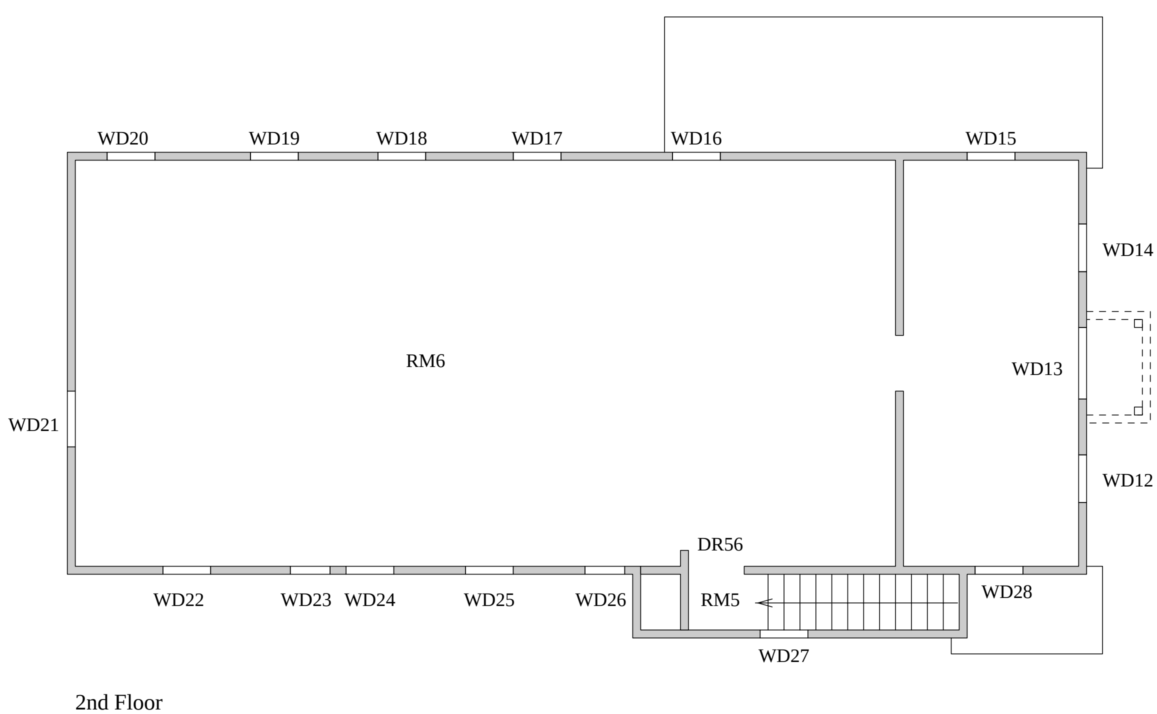

Figure 6.1 B: *Layout of the 2nd Floor*

Table 6.1: *Flow Coefficients, Exponents and Calculated Air Flows for the Flow Paths*				
#	Flow Path Name	Flow Coefficient [kg / s @ 1Pa]	Flow Exponent [-]	Flow Rate [kg /h]
1	DR23	0.0833	0.53	2.68E02
2	RM26	0.0044	0.60	2.82E01
3	DR12	0.0691	0.54	-1.67E02
4	DR13	0.0860	0.53	1.72E02
5	RM16	0.0044	0.60	2.50E01
6	DR34	0.0782	0.54	.98E02
7	RM36	0.0109	0.60	5.61E01
8	DR45	0.0744	0.55	2.20E02
9	RM46	0.0130	0.60	3.78E01
10	DR56	0.0784	0.56	-2.85E00
11	RM1E	0.0055	0.78	-2.91E01
12	RM2E	0.1200	0.50	-4.64E02
13	RM3E	0.0029	0.61	-1.57E01 14
14	RM3N	0.0055	0.61	4.31E01 15
15	RM3W	0.0078	0.61	5.85E01
16	RM4E	0.0093	0.51	-5.94E01
17	RM4N	0.0029	0.51	1.66E01
18	RM4W	0.0024	0.51	1.30E01
19	RM4S	0.0122	0.51	6.97E01
20	RM5W	0.0419	0.52	1.87E02
21	RM5S	0.0074	0.52	3.58E01
22	RM6E	0.0113	0.67	-1.02E02
23	RM6N	0.0046	0.67	2.25E01
24	RM6W	0.0074	0.67	3.22E01
25	RM6S	0.0046	0.67	2.25E01 26
26	RM6R	0.0347	0.67	1.97E02

6.2 Input File

The input file contains all the data that is necessary for the calculations. To make the procedure clearer, in this example, the amount of data is limited. Section 5 'Input Data Description and Input Format' is a complete description of the input. It is recommended that these two texts are read simultaneously.

We will now go through the input file step by step and explain each one of the input data - where they come from and what they are used for. The complete input file is shown on the following pages.

&-PR-IDENtification (Optional)
1. PROBLEMNAME is a short description of the example.

2. VERSIONNAME is the date when the example was created. It could also be any other identification of the current run.

&-PR-OUTPut options (Mandatory)
A **VENT**ilation calculation is desired and is therefore filled out. The program will now calculate the air flow rates and pressures throughout the building. Several additional options are possible.

&-CR CRACK (Mandatory)
Under this heading, all the flow links of type **CR**acks are filled out. This is one of several Air Flow Components (see Section 4.6). A crack may represent the leakage path through a wall, a door, or a window. The data are identical with the data given in Table 6.1. Our data are given as flow coefficients and exponents and are filled out accordingly. The data originate from a fitting procedure of the parameters (Cm) and (EXP n) in the equation below to the measurement data.

Each flow path is given two lines in the form:

```
name 1 name 2          name 3
Cm                     Exp n
```

where	name 1	name of air flow component, here *CR for crack
	name 2	your own identification of the flow path
	name 3	description of the flow path
	Cm	flow coefficient [@ {kg}/s {1 Pa }@]
	Exp n	flow exponent [-]

&-NET-ZONes (Mandatory)
The building is divided into zones which often are the real physical rooms. This section should be filled out with information related to the zones such as volume and temperature.

&-NET-EXTernal node data (Optional)

Here, every external node, i.e. nodes around the building, is associated with a facade number.

&-NET-LINKS (Mandatory)
Every crack connects two zones or a zone and an external node. A zone with a negative sign represent an external node, i.e., constant outside condition (pressure, temperature, concentration). HEIGHT given for a zone is given in reference to the reference level in the zone, while the HEIGHT for an external node is in reference to the building reference level.

&-CP-BUILding reference height for Cp data (Optional)
The reference height of the building concerning the given Cp-values should be given here. It is usually the level of the roof of the building.

&-CP-VALUes (Optional)
1. DATASET NAME is a reference name of the Cp-data set. The wind directions (N=0°, E=90°,...) and the facade numbers are given in Figure 6.1. The Cp-value relates the actual wind pressure to the wind pressure at the building reference height.

2. The appropriate Cp-values are given for different facades and wind directions.

&-ENV-WIND and meteo data related parameters (Optional)
Data needed to calculate the wind velocity at the building reference height. The plan area density describes the obstruction density of surrounding buildings. The wind profile exponent of 0.32 correspond to a heterogeneous surface with structures taller than one story.

&-SCH-METeo data (Optional)
1. DATASET NAME is a reference name of the meteo data set.

2. Outdoor meteo data are given here. The TIME is used to schedule the meteo data.

Output File
In Section 7 one finds a complete description of the Output. It composes the air flow rates and the zone pressures. The results obtained for the example run are shown in Table 6.1.

7. Output

7.1 Input Data

7.2 Calculated Data

The current version of COMIS has the following output options :

COMIS Output File (cof)
Formatted User Output File (userf)
Unformatted User Output File (useru)

7.2.1 COMIS Output File (cof)

The COMIS OUTPUT FILE is the main destination for the data calculated by COMIS. What kind of data is written to the COF is specified by keywords in the &-PR-OUTP section of the input file. It is also possible to direct all screen output of COMVEN to the COF by setting a flag in the COMIS.SET file.

7.2.1.1 Ventilation Output

This data is written to the COF when the 'VENTout' keyword is present in the &-PR_OUTP section.

An example of the output from the sample input file follows. The output has been significantly reduced :

```
  NO ventilation ERRORS REPORTED

1 room
 nonlinear solution found after  4 iterations

Zone-ID  pressure (Pa)   totalflow (kg/s)    imbalance
(kg/s)

   1       -108.28           2.341E-02         -2.959E-11


link           from        to         Tlink    Dp-link    fma1
               fma2

nr  name type typ  name   typ  name C    Pa      kg/s   kg/s


1  lin1   CRn  ex   1    zn   1    20.0 -5.48     .0   2.3E-
   02

2  lin2   CRn  zn   1    ex   3    20.0 -3.81     .0   2.3E-
   02
```

Description of the example:
" NO ventilation ERRORS REPORTED " means that the output is OK.
"Model name: 1 room" is the name given in the input file.
"nonlinear solution found after 4 iterations" tells how many iteration steps were necessary to solve the network.

Description of zone output.
"Zone-ID 1" This is the output for the first zone.
"pressure (Pa) -108.28" This is the pressure at the zone reference plane.
"totalflow(kg/s) 2.341E-02" this is the total flow that flows through the zone. In fact it is half the sum of the absolute values of all flows to and from this zone.
"imbalance (kg/s) -2.959E-11" this is the error in the flow balance for this zone. For an ideal solution this error would be zero. The value must meet the tolerances given under &PR-CONTrol.

Description of link output.
"nr" is the sequence number of the link.
"name" is the name of the link given in the input file.
"type" is the name of the AFC from which the coefficients have been used for this link.
"from-to" are the names of the zones which are connected by this link. Instead of zones the connected items may be "special pressures" or "facade elements". Therefore the linked items are preceded by "zn" for zone, "sp" for special pressure or "ex" for (external) facade element number.
"Tlink" is the average temperature of the air flowing through the link "Dp-link" the pressure difference across the link.
"fma1" is the outgoing flow rate. fma1 and fma2 are also used to indicate a two-way flow at large openings.
"fma2" is the incoming flow rate.

7.2.1.2 Ventilation Output '2VENT'

This data is written to the COF when the '2VENTout' key-word is present in the &-PR OUTP section, which means that the second output routine is selected.

The output follows a different format.

An example of the output from the sample input file follows. Here follow the most important differences compared with the 'VENT'output option (output has been reduced):

```
  Meteo: Vmeteo=3.m/s Direction90.deg temp=20.C 12-iterations
Solver=5
  Vbuilding=2.894m/s
 Cpnr   Cp      Pa windpressure
   1      -0.500      -2.521
   2       0.600       3.025
   3      -0.500      -2.521
   4      -0.450      -2.269
   5      -0.500      -2.521
 ----link------- --from-- ---to-- Tlink  Dp-link from->to
velocity
 nr  ID   type   typ name typ name C     Pa       dm3/s     m/s
 ------------------------------------------------------------
-----
   1 1    CRDR23 z RL1    z RL2    20. -1.99E-3 -2.576E+0 -
0.04848
   2 2    CRRM26 z RL2    z RA6    20.  1.52E+0  4.721E+0  1.766
   3 3    CRDR12 z RL1    z RL2    20. -1.99E-3 -2.008E+0 -
0.04587
   4 5    CRRM16 z RL1    z RA6    20.  1.52E+0  4.717E+0  1.764
   5 11   CRRM1E z RL1    e 2      20. -1.07E-2 -1.328E-1 -
0.04501
   6 12   CRRM2E z RL2    e 2      20. -8.66E-3 -9.304E+0 -
0.1198
   7 25   CRRM6S z RA6    e 1      20.  3.83E+0  9.418E+0  3.56

 Zone-ID pressure totalflow infiltration ventilation imbalance
temp
          Pa        dm3/s     dm3/s       dm3/s      dm3/s     C
 ------------------------------------------------------------
-----
  RL1    -2.779     4.717     0.1328      4.584     -0.000185
20.
  RL2    -2.777     9.304     9.304       0.        -0.000332
20.
  RA6   -36.347     9.437     0.          9.439     -5.29E-08
20.

 Total infiltration   =   9.437025     dm3/s
 Total air change rate=  6.5333255E-02 1/h
 Total building volume=   520.0000     m3
```

```
 -----Room---- -------flow rates-----  ---air change rates------
-
  name   volume  inf    vent   total   inf      vent     total
           m3   dm3/s   dm3/s  dm3/s   1/h       1/h       1/h
 RL      104.0   9.4     0.0    9.4    0.3       0.0       0.3
 RA      416.0   0.0     9.4    9.4    0.0       0.1       0.1
  ..................................................
```

The output first states the meteo conditions and the local wind velocity at the building (Vbuilding), followed by the wind pressures.

In the Link part the velocity in the net area of the openings is given. For industrial ventilation these are the flows that can be measured in the ventilation openings, which enables a direct fit of the model.

The Zone part shows total flow through the zone, the infitration flow which enters from outside and the ventilation flowrate which enters from other zones.

A summary is given for the whole building.

At last follows a grouped output for all zones for which the first two characters of the zone ID are the same, here RLx and RA for rooms at Level1 and rooms at the attic level. At an efficient choice of zone-ID's one could get here the totals per department or for instance per floor or module of the building. This will make it easier to analyse the output

7.2.1.3 Pollutant Output

This data is written to the COF when the 'POLout' keyword is present in the &-PR_OUTP section. If no pollutant output is specified the calculation of concentrations is skipped also.

An example of the data written to the COF follows:

```
Pollutant transport output   ==========================
   Pollutant No.  1  (  FORMALDEHYD  )

   Zone-ID     Source         Sink      Concentration
          kg/s      kg/s      kg/kg
  ----------------------------------------------------------
-------------------------
      1        0.        0.        0.004709
      2        0.        0.        0.01347
      3        0.01      0.001          0.0236
```

Description of the example:

"Pollutant No. 1 (FORMALDEHYD)" gives the number and name of the pollutant for the following table (there is one table for each pollutant).

"Source" is the current source strength for the pollutant in the given zone.

"Sink" accordingly

"Concentration" is the result of the calculations in the pollutant transport model. It shows the concentration of the pollutant in the zone.

7.2.2 User Output File (uof)

Besides the COMIS output file (cof) the user has the option to create his own output file. This user output file allows to manipulate the output (e.g., adding flows) and to reduce output to absolute essential data. This output file can be a formatted or an unformatted output file. The output routine has to be modified by the user according to his needs.

To use this option the user must have access to the source code and you must have a FORTRAN COMPILER, because after the modification the subroutine must be re-compiled and linked with the program. Two examples how to modify and to compile the output subroutine follow.

7.2.2.1 Formatted User Output File (USERF)

To work with a user created output file (formatted), it is necessary to modify the subroutine according to the user's needs. Therefore, the subroutine 'UserOutF(file)' has to be loaded into a text editor. The subroutine is part of the module 'comv-user.f'. The following example gives an overview over the data that can be used and what the subroutine has to look like.

```
C****************************************************************
**
      SUBROUTINE UserOutF(file,time,interval,timestring)
C****************************************************************
**
C*    1991oct28   ENVUXR
C*    This subroutine allows the user to create his own
formatted output file.
C*    A list with names of variables that are available follows.
C*    Following the data description an example is given.
C*    'UserOutF' is part of the module 'comv-user.f'.
C****************************************************************
**
C
C parameter/dimension (dimensions are set in the include files)
C nw    =  number of wind pressure points (max=maxw)
C nz    =  number of zones (max=maxz)
C nl    =  number of links (max=maxl)
C maxc  = 5     ( max. number of concentrations,different gases
)
C maxcp = 1000( max. number of Cp*number of directions
C in the input file )
C maxl =  300   ( max. number of links
C maxw =  200   ( max. number of wind pressure points )
C maxz =  200   ( max. number of zones )
C
C name/dimension   unit      type          description
```

```
C ========================================================
C Pz(maxz)          Pa     double     Pressure per Zone
C FMB(maxz)        kg/s    double     Flow Mass Balance per
zone.
C Tz(maxz)         C       real       air temperature per zone.
C                                      Gradients are possible
with
C                                      Layers.
C Xhz(maxz)        kg/kg   real       absolute moisture content
per
C                                      zone. Gradients are
possible
C                                      with Layers.
C FT(maxz)         kg/s    real       total incoming flow per
zone
C C(maxc,maxz)     kg/kg   real       concentration of gasses
per
C                                      zone.
C                                      Gradients are possible
with
C                                      Layers
C                                      but only for the first gas
C                                      (1,..)
C cout(maxc,nw)     kg/kg  real       concentration of gasses
per
C                                      outside
C                                      facade element (parallel
with
C                                      the
C                                      cp values, no gradients)
C Source(maxc,maxz)  kg/s  real       pollutant source strength
per
C                                      zone
C Sink  kg/s               real       pollutant sink strength
per zone
C FV(maxl)            m3/s real       mass flow vector
C Tl(maxl)              C   real       temperature per link
C FV2(2,maxl)        kg/m3 real       Flow vector with the 2-way
flows
C                                      per link
C Cp(maxcp)           [-]  real       wind pressure coefficient
C Pwind(maxw)          Pa  real       wind pressure per wind
pressure
C                                      point
C Vmet                 m/s real       meteo velocity
C Tmet                   C  real       meteo air temperature
C Xhmet              kg/kg real       absolute humidity at meteo
C FromTo(2,maxl)      [-]  integer    the both zones a link
connects,
C FromTo(1,I)                          is the zone link I departs
C From. FromTo(2,I)                    is the zone it goes
C To. Negative flows go the other way, but are dealt properly
C i.e. at summing the flows per zone
C LStat(maxl)   [-]   integer         Link Status:
C Lstat         From  TO
```

```
C 0             zone  zone
C 1             ex    zone
C 2             spec  zone
C 3             zone  ex
C 4             ex    ex
C 5             spec  ex
C 6             zone  spec
C 7             ex    spec
C 8             spec  spec
C Cname[-] char  name of pollutant
C MM  g/mol real  molar mass of the pollutant
C Nconc[-] integer    number of concentration
C---------------------------------------------------------------
-
C Parameter
C file [-] integer    unit number for data output file
C time [sec]    double     current simulation time
C interval [sec] integer    time interval
C timestring    datetime   string     current date/time in
string
C***************************************************************
**
C the following include files have to be part of this subroutine
:
      INCLUDE 'comv-inp.inc'
      INCLUDE 'comvuni.inc'
      INTEGER file,interval
      DOUBLE PRECISION time
      CHARACTER*18 timestring
C This example shows how to print the pressure per zones (Pz)
and the average
C of all zone pressures. The data is written into the user
output file 'file'.
C============================================================
C
C local variables determined by user :
C => pz_cumulated consists of the sum of all pz(i)
C => pz_average is the result of pz_cumulated divided by i
      real pzcumulated
      real pzaverage
      pzcumulated = 0.
C this command writes the results in one row, whereby the number
of the results
C is dependent on the 'number of zones' (nz=>   max=maxz)
      WRITE(file,111)
     &      'RESULT PRESSURE PER ZONE # ' ,(pz(i), i=1,nz)
C this command calculates the average pressure :
      DO 10 i=1, nz
          pzcumulated = pzcumulated + pz(i)
10    CONTINUE
      pzaverage = pzcumulated/i
C this command writes the average pressure in the output file :
      WRITE(file,222)
     &      'AVERAGE PRESSURE WITH',i,'ZONES ==>> ',pzaverage
111   FORMAT(A,10F8.2)
```

```
222   FORMAT(I4,F8.2)
      RETURN
      END
C*****************************************************************
**
```

After the modification, COMIS has to be re-compiled. If the command 'Make' is used, only subroutines in which the source code had been changed will be compiled. There are two ways to call the user specified output routine in COMIS :

in the SET.FILE with the keyword USEROUTF name

in the COMIS-command line with the option '-uf name'

For more details, please see chapter 2.2.

7.2.2.2 Unformatted User Output File (USERU)

To work with a user created output file (unformatted), it is necessary to modify the subroutine according to the user's needs. Therefore, the subroutine 'UserOutU(file)' has to be loaded into a text editor. The subroutine is part of the module 'comv-user.f'. The following example gives an overview over the data that can be used and what the subroutine has to look like.

```
C***************************************************************
**
          SUBROUTINE UserOutU(file,time,interval,timestring)
C***************************************************************
**
C*    1991oct28   ENVUXR
C*    This subroutine allows the user to create his own
unformatted output file.
C*    A list with names of variables that are available follows.
C*    Following the data description an example is given.
C*    'UserOutU' is part of the module 'comv-user.f'.
C***************************************************************
**
C parameter/dimension (dimensions are set in the include files)
C nw      =  number of wind pressure points (max=maxw)
C nz      =  number of zones (max=maxz)
C nl      =  number of links (max=maxl)
C maxc    = 5  ( max. number of concentrations,
C               different gases )
C maxcp   = 1000     ( max. number of Cp*number of
C               directions in the input file )
C maxl    =  300     ( max. number of links
C maxw    =  200     ( max. number of wind pressure points )
C maxz    =  200     ( max. number of zones )
C end of dimension ***
C
C name/dimensionunit type description
C =========================================================
C Pz(maxz) Pa   double    Pressure per Zone
C FMB(maxz)     kg/s double    Flow Mass Balance per zone.
C Tz(maxz) C    real air temperature per zone.
C                    Gradients are possible with
C                    Layers.
C Xhz(maxz)     kg/kg real absolute moisture content per
C                    zone. Gradients are possible
C                    with Layers.
C FT(maxz) kg/s real total incoming flow per zone
C C(maxc,maxz)  kg/kg real concentration of gasses per zone.
C                    Gradients are possible with Layers
C                    but only for the first gas C(1,..)
C cout(maxc,nw) kg/kg real concentration of gasses per outside
C                    facade element (parallel with the
```

```
C                       cp values, no gradients)
C Source(maxc,maxz)   kg/s  real  pollutant source strength per
zone
C Sink     kg/s  real  pollutant sink strength per zone
C FV(maxl) m3/s  real  mass flow vector
C Tl(maxl) C     real  temperature per link
C FV2(2,maxl)   kg/m3 real  Flow vector with the 2-way flows
C                       per link
C Cp(maxcp)     [-]   real  wind pressure coefficient
C Pwind(maxw)   Pa    real  wind pressure per wind pressure point
C Vmet     m/s   real  meteo velocity
C Tmet     C     real  meteo air temperature
C Xhmet    kg/kg real  absolute humidity at meteo
C FromTo(2,maxl)[-]   integer    the both zones a link connects,
C                       FromTo(1,I) is the zone link I departs
C                       From. FromTo(2,I) is the zone it goes
C                       To. Negative flows go the other
C                       way, but are dealt properly
C                       i.e. at summing the flows per zone
C LStat(maxl)   [-]   integer         Link Status:
C                       LStat From  TO
C                       0     zone  zone
C                       1     ex    zone
C                       2     spec  zone
C                       3     zone  ex
C                       4     ex    ex
C                       5     spec  ex
C                       6     zone  spec
C                       7     ex    spec
C Cname    [-]   char  name of pollutant
C MM       g/mol real  molar mass of the pollutant
C Nconc    [-]   integer    number of concentration
C-----------------------------------------------------------------
--
C Parameter
C file     [-]   integer    unit number for data output file
C time     [sec] double     current simulation time
C interval [sec] integer    time interval
C timestring    datetime   string     current date/time in
string
C*****************************************************************
**
C the following include files have to be part of this subroutine
:
           INCLUDE 'comv-inp.inc'
           INCLUDE 'comvuni.inc'

           INTEGER file,interval
           DOUBLE PRECISION time
           CHARACTER*18 timestring

C This example shows how to print the pressure per zones (Pz)
and the average
C of all zone pressures. The data is written into the user
output file 'file'.
```

```
C============================================================
C
C local variables determined by user :
C => pz_cumulated consists the sum of all pz(i)
C => pz_average is the result of pz_cumulated divided by i
           real pzcumulated
           real pzaverage

           pzcumulated = 0
C this command writes the pressure per zones into the output
file :
           Write(file) ( pz(i), i=1,nz)
C this command calculates the average per pressure :
           DO 10 i=1, nz
               pzcumulated = pzcumulated + pz(i)
10         CONTINUE
           pzaverage = pzcumulated/i

C this command writes the average into the output file :
           WRITE(file) pzaverage

           RETURN
           END
C***************************************************************
**
```

After the modification, COMIS has to be re-compiled. IF the command 'Make', is used, only subroutines, which source code had been changed will be compiled. There are two ways to call the user specified output routine in COMIS :

in the SET.FILE with the keyword 'USEROUTU name'

in the COMIS-command line with the option '-uu name'.

For more details, please see chapter 2.2

7.3 Error Messages

All routines in COMVEN report their errors to the COMIS.CER file. Most errors will probably occur due to improper input or mistakes in the network of links.

8. Acknowledgments

This publication was supported by the following organizations:

Carl-Duisberg-Gesellschaft e.V., Koeln, Federal Republic of Germany; National Scientific Research Center (C.N.R.S.), 75700 Paris, France; French Agency for Energy Management (A.F.M.E.), 75015 Paris, France; Dept. for Environment Science and Technology, Politecnico di Torino, Italy; S.I.V. (Societa' Italiana Vetro) S.p.a., Italy; SECCO S.p.a. , Treviso, Italy; Netherlands Organization for Applied Scientific Research, Division of Technology for Society, Department of Indoor Environment, Delft, The Netherlands; Miyagi National College of Technology, Japan; Tohoku University, Japan; Harbin Architectural and Civil Engineering Institute, Harbin, People's Republic of China; CIEMAT-IER, Madrid, Ministerio de Educacion y Ciencia, Madrid, as well as Escuela Superior de Ingenieros Industriales, Sevilla, Spain; The Swedish Council for Building Research, Stockholm, Sweden; Ecole Polytechnique Federale de Lausanne, as well as Eidgenoessische Materialpruefungs- und Forschungsanstalt (EMPA), Abt. 175, Projekt ERL, NEFF, Switzerland; Air Infiltration and Ventilation Centre, Warwick, UK; and by the Assistant Secretary for Conservation and Renewable Energy, Office of Building and Community Systems, Building Systems Division of the U.S. Department of Energy under Contract No. DE-AC03-76SF00098.

9. Disclaimer

This document was prepared as an account of work sponsored by the above mentioned agencies. None of these agencies, nor the Regents of the University of California, nor any of their employees or guests, makes any warranty, express or implied, or assumes any legal liability or responsibility for the accuracy, completeness, or usefulness of any information, apparatus, product or process disclosed, or represents that its use would not infringe privately owned rights. References herein to any specific commercial products, process, or service by its trade name, trademark, manufacturer, or otherwise, does not necessarily constitute or imply its endorsement, recommendation, or favoring by any of the sponsoring agencies or the Regents of the University of California. The views and opinions of authors expressed herein do not necessarily state or reflect those of the sponsoring agencies, or the Regents of the University of California, and shall not be used for advertising or product endorsement purposes.

10. Errata

Appendices

Comis User's Guide Input File (.CIF)   comisman.cif 91-02-18
#==
==

&-PR-IDENtification

1. Problemname

This is the input example for the COMIS User's Guide.
The building is: 178 Richmond Field Station,
University of California, Berkeley, California, USA.

2. Versionname

October 13, 1993

&-PR-OUTPut options

Output Option Keywords: One keyword per line only
Keywords may be preceded by NO

VENT:ilation POL:utantHEAT:flow
CONC:entrations
INPUT echo
DEFAULT echo
SET echo
SCHED:time< time>
START:time< time> [CONT| REUSE] STOP:time< time> [KEEP]|

Graphical Output Options: Define data to be Stored:
PZ-S { Zones} = Pressure | FL-S { Links}   = Flow
TZ-S { Zones} = Temperature | TL-S { Links}   = Temperature
MZ-S { Zones} = Moisture | SL-S { Links}   = Status
FZ-S { Zones} = Flow | HU-G = Humidity
VE-G = Velocity | TE-G = Air Temp.
Cn-S { Zones} = Concentr. | Pn-S { Zones}   = Poll. Str.
Sn-S { Zones} = Poll. Sink | WP-S { Points} = Windpr.
|for Gas n (1< = n < =5)

To define graphs: replace -S with -T (Table entry)

STARTtime 1991jun15_10:00
STOPtime 1991jun15_11:00
VENT

&-PR-CONTrol parameters--- OPTIONAL DATASECTION ---

1.	Under Relax-ation Factor	Tolerances absolute EpsFA	Relative EpsFR	CORR*JAC(i,i) EpsCJ	Start Number of Iterations	Link Flow Pressure Laminar Flow DifLim
	[-]	[kg/s]	[-]	[kg/s]		
1.		1.E-06	1.E-05	0.	1	1.E-12

2.	use old Pressures	No Pressure Initialization	Solver Selector	Max Number of Iterations allowed
0=	Zero Pressures	0=Lin.initial.	0=optimum relax COMIS	
		1=No initial	1=Newton (with given Relax)	
1=	use Previous		2=Newton Steffensen	
			3=Walton Steffensen	
			4=One avg. Steffensen	
	UseOPz		5=Walton 2 fixed relax.fact	
		NoInit	SlvSel	Miter
	[-]	[-]	[-]	[-]
0		0	5	500

&-NET-AIR flow components

```
# Allowed prefixes are:     *CR   *FA  *DS  *DF  *F1   *F2   *F3  *F4    *WI   *TD
#                           |     |     |     |     |____________________| |     |
#                           crack |    duct   |  flow-controllers          |     testdata points
#                                 fan         duct-fitting                 window (openable)
# keep the KEYWORDs &-CR,...,&-TD in this part &-NET-AIR
```

&-CR CRACK

1.	Cs	Exp n	Length	Wall Thickness	Properties U-Value
	(kg/s@ 1Pa)	(-)	[m]	[m]	[W/m2 K]

2.	Filter 1	Filter 2	Filter 3	Filter 4	Filter 5
	(-)	[-]	[-]	[-]	[-]

*CRDR23 Internal door
0.0833 0.53
0.0

*CRRM26 Vertical Leakage
0.0044 0.60
0.0

*CRDR12 Internal door
0.0691 0.54
0.0

*CRDR13 Internal door
0.0860 0.53
0.0

*CRRM16 Vertical Leakage
0.0044 0.60
0.0

*CRDR34 Internal door
0.0782 0.54
0.0

*CRRM36 Vertical Leakage
0.0109 0.60
0.0

*CRDR45 Internal door
0.0744 0.55
0.0

*CRRM46 Vertical Leakage
0.0130 0.60
0.0

*CRDR56 Vertical Leakage
0.0784 0.56
0.0

*CRDR6R Roof
0.0347 0.67
0.0

*CRRM1E Outer wall
0.0055 0.78
0.0

*CRRM2E Outer wall
0.1200 0.50
0.0

*CRRM3E Outer wall
0.0029 0.61
0.0

*CRRM3N Outer wall
0.0055 0.61
0.0

*CRRM3W Outer wall
0.0078 0.61
0.0

*CRRM4E Outer wall
0.0093 0.51
0.0

*CRRM4N Outer wall
0.0029 0.51
0.0

*CRRM4W	Outer wall
0.0024 0.51
0.0

*CRRM4S	Outer wall
0.0122 0.51
0.0

*CRRM5W	Outer wall
0.0419 0.52
0.0

*CRRM5S	Outer wall
0.0074 0.52
0.0

*CRRM6E	Outer wall
0.0113 0.67
0.0

*CRRM6N	Outer wall
0.0046 0.67
0.0

*CRRM6W	Outer wall
0.0074 0.67
0.0

*CRRM6S	Outer wall
0.0046 0.67
0.0

*CRRM6R	Roof
0.0347 0.67
0.0

&-NET-ZONes

Zone No (-)	Name [-]	Temp [oC]	Ref. Height [m]	Vol [m3]	Abs. Hum [gr/kg]	Schedule Name [T./H..]
1	RM1	20.	0.5	52.		
2	RM2	20.	0.5	52.		
3	RM3	20.	0.5	130.		
4	RM4	20.	0.5	156.		
5	RM5	20.	0.5	26.		
6	RM6	20.	3.2	416.		

&-NET-EXTernal node data --- OPTIONAL DATASECTION ---

External Node No	Facade Elem No	Outside Conc Factor
(-)	(-)	[-]
1	1	0.
2	2	0.
3	3	0.
4	4	0.
5	5	0.

&-NET-LINks

Link	Type	Zone No		Height		Own	Act.	3Dflow or	Schedule Name(5char.)	
									T-Junct.	Ref.Link
No	Name	From	To	From	To	Height	Val.	Press	No	Angle
(-)	(-)	(-)	(-)	[m]	[m]	[m]	[-]	[Pa]	[-]	[deg]

1.	CRDR23	2	3		
2.	CRRM26	2	6	2.7	0.0
3.	CRDR12	1	2		
4.	CRDR13	1	3		
5.	CRRM16	1	6	2.7	0.0
6.	CRDR34	3	4		
7.	CRRM36	3	6	2.7	0.0
8.	CRDR45	4	5		
9.	CRRM46	4	6	2.7	0.0
10.	CRDR56	5	6	2.7	0.0
11.	CRRM1E	1	-2	1.5	2.0
12.	CRRM2E	2	-2	1.5	2.0
13.	CRRM3E	3	-2	1.5	2.0
14.	CRRM3N	3	-3	1.5	2.0
15.	CRRM3W	3	-4	1.5	2.0
16.	CRRM4E	4	-2	1.5	2.0
17.	CRRM4N	4	-3	1.5	2.0
18.	CRRM4W	4	-4	1.5	2.0
19.	CRRM4S	4	-1	1.5	2.0
20.	CRRM5W	5	-4	1.5	2.0
21.	CRRM5S	5	-1	1.5	2.0
22.	CRRM6E	6	-2	1.5	4.7
23.	CRRM6N	6	-3	1.5	4.7
24.	CRRM6W	6	-4	1.5	4.7
25.	CRRM6S	6	-1	1.5	4.7
26.	CRRM6R	6	-5	1.5	4.7

&-CP-BUILding reference height for Cp data --- OPTIONAL DATASECTION ---

Height
(m)
10.0

&-CP-VALUes --- OPTIONAL DATASECTION ---

1. Dataset Name

Richmond-Cp

2.	Facade	Winddirection				(first line)				
	Elemno	Cp	Values			(second and following lines)				
*	(-)	[deg]	[deg]	[deg]	(deg)	[deg]	[deg]	[deg]	[deg]	[deg]

*	0.	90.	180.	270.
1	-0.45	-0.5	0.6	-0.5
2	-0.5	0.6	-0.5	-0.45
3	0.6	-0.5	-0.45	-0.5
4	-0.5	-0.45	-0.5	0.6
5	-0.5	-0.5	-0.5	-0.5

&-ENV-WINd and meteo related parameters --- OPTIONAL DATASECTION ---

1.	Ref. Height	Altitude
	for Wind Speed (m)	Meteo Station (m)
10.		50.

2.	Wind		Wind	
	Direction Angle	Plan Area Density	Velocity Profile Exponent	Surrounding Buildings Height
	(deg)	(-)	(-)	(m)
0.		0.2	0.32	6.
90.		0.2	0.32	6.
180.		0.2	0.32	6.
270.		0.2	0.32	6.

&-SCH-METeo data --- OPTIONAL DATASECTION ---

#---------------------
METEO DESCRIPTION
#---------------------

_ _ _ _ _ _ _ _ _ _ _ _ _ _ _ _ _

1. Dataset Name

Richmond-Meteo

2. Time	Wind		Temperature	Humidity	Barometer Pressure
	Speed	Direction			Absolute
(-)	(m/sec)	(deg)	(oC)	[g/kg]	[kPa]
1991jun15_ 10:00	3.	90.	20.	10.	101.3

Alphabetical List of COMVEN Keywords and Their Status

Keyword	Description	Status
CP-BUILding	Building reference height for Cp	functional
CP-VALUe	Cp value	functional
CPR-BUIlding	Building rough outside dimensions	functional
CPR-FACade	Facade element definition	functional
CPR-WINd	Wind direction data	functional
ENV-BUIlding	Building related data	functional
ENV-WINd	Wind and meteorological data	functional
NET-AIRflow	Air flow components:	
CR	Crack	functional
DF	Duct fitting	functional
DS	Straight duct	functional
F1,F2,F3,F4	Flow controllers	functional
FA	Fan	functional
TD	Test data component	unknown
TRANSITion	Duct transition	non-functional
WI	Window/door	functional
NET-EXTernal	External node definition	functional
NET-HVAc	HVAC network	non-functional
NET-LINks	Links	functional
NET-ZL	Zone layers	unknown
NET-ZONes	Zone definition	functional
NET-ZP	Zone pollutants	unknown
NORM-CRack	Normalize crack temperature, pressure and humidity ratio	functional
OCCUPANt	Occupant description	unknown
POL-DEScription	Pollutant description	functional
PR-CONTrol 1,3,4,5	Problem control parm definition	functional for solvers
PR-IDENtification	Problem Identification	functional
PR-UNITs	Input/Output Unit conversions	functional
PR-OUTPut	Output Option header:	
DEFAULT	Print defaults to .COF file	functional
HEAT:flow	Calculation with heatflow module	non-functional
INPUT	Input file is printed to .COF file	functional
CONC:entrations	Concentrations of specified pols.	functional
POL:lutant	Pollutant flow is modeled	functional
SCHED:time	Start time of processing schedules	functional
SET	SET-file is printed into .COF file	functional
START:time	Start time of simulation	functional
STOP:time	Stop time of simulation	functional
VENT:ilation	Calculation with COMVEN	functional
SCH-FAN	Fan schedule	functional
SCH-HUM	Humidity schedule	functional
SCH-LINk	Link exchanger schedule	non-functional
SCH-MAIn	Main schedule	non-functional
SCH-METeo	Meteorological (weather) schedule	functional

SCH-MULti	Multi-schedules	functional
SCH-OCCupant	Occupant schedule	unknown
SCH-POLlutant	Outdoor pollutant schedule	unknown
SCH-SINk	Pollutant sink schedule	unknown
SCH-SOUrce	Pollutant source schedule	unknown
SCH-TEMperature	Temperature schedule	functional
SCH-WINdow	Window schedule	functional